A Survey of Numerical Models for Wind Prediction

D. Schonfeld
DSN Engineering Section

A literature review is presented of the work done so far in the numerical modeling of wind flows. Pertinent computational techniques are described, as well as the necessary assumptions used to simplify the governing equations. A steady-state model is outlined, based on the data obtained at the Deep Space Communications complex at Goldstone, California.

I. Introduction

In recent years, due to the increasing cost of oil and other fossil fuels, wind-generated electricity has been actively sought as a possible alternate and renewable energy source. At the present time, there exists a national R&D program to test the engineering and economic feasibility of wind-driven turbines.

In determining the site and construction of such turbines, a necessary starting point is the collection of data on wind speed, direction and duration. Since this data collection is limited in scope due to resource restrictions, alternative means must be sought. This siting problem can be stated as follows: for a certain area, given the topography of the terrain and given the wind direction and speed at one or more of the area boundaries, it is desired to predict the wind distribution at all points within the stated boundaries.

The solution of the problem can be quite complex due to *three* main sources of difficulty. First, the problem is complicated by the topography of the terrain, which may be highly irregular, requiring a large amount of data to describe it. Second, the wind data, given at one or more of the boundaries, is highly irregular with respect to temporal variation in magnitude and with changes of direction. Therefore, there are difficulties related to processing the large amounts of data necessary for describing these boundaries. Third, and of equal importance, is the fact that the fluid dynamics and energy equations describing the wind flow are too complicated to be solved by analytic means in closed form. Because of these reasons, any analysis for the prediction of wind velocities over an arbitrary terrain must be treated by numerical, i.e., computer methods.

This article is written with the aim of introducing the reader to various aspects of wind prediction research. The first section presents the results of a literature search made during June – December 1979 into this subject and it also describes the past DSN activities in this field.

The next four sections discuss the physical and computational aspects of pertinent numerical modeling. The article concludes with suggestions for future work.

II. Literature Review

During the period from June – December 1979, a review of the literature pertaining to wind velocity predictions has been conducted in the Energy Group of the DSN Engineering

Section. It became apparent that most of the research effort in this area had been concentrated at only a few laboratories and universities. The results obtained at these centers are presented below.

A. Work Performed at Los Alamos Scientific Laboratory

The Los Alamos Scientific Laboratory (LASL) has a strong tradition of computer modeling dating back to the 1940s. The laboratory possesses some of the most modern computers available and an excellent technical staff. This combination has produced several classic techniques for fluid flow modeling such as the Marker And Cell (MAC) (Ref. 1) and the Particle In Cell (PIC) methods (Ref. 2). In recent years, LASL has published a number of papers dealing with computer programs that model the spread of wind-borne pollutants (Refs. 3-5). These programs are characterized by (1) solutions of the complete, unsteady Navier-Stokes equations, and (2) the application of these solutions to flows past obstacles with a regular geometry, e.g., cubes and rectangular blocks. The results obtained show very good agreement with experimental observations of pollutant dispersion.

A penalty paid for obtaining such good results is the high cost incurred by operating these programs. Nearly all the storage of the 64,000-word fast core memory of a CDC – 7600 computer is taken up by these codes. Within these programs, there are approximately 3300 computational cells, each of them requiring approximately 1.5 sec of computation time per cycle; this means significantly more than 1 hour of computation time for each cycle. Some of the simpler programs developed at LASL (such as SOLA) still require large amounts of computer time and cost. A rough estimate (Ref. 10) shows that these costs are proportional to (N^4), where N is the number of mesh cells into which the length dimension is divided, and with (K^3), where K is proportional to the distance that the mesh boundaries must be kept away from the flow region of interest. Therefore, computational costs can be kept within some reasonable limits by accepting the minimum resolution that will still yield accurate results, that is reducing N and providing realistic boundary conditions, which means minimizing K. It was shown (Ref. 6) that these cost penalties still remain high even when the physical assumptions of the problem are slightly relaxed.

B. Work Performed at Lawrence Livermore Laboratory

A second center which is involved in research on wind field prediction is the Lawrence Livermore Laboratory (LLL). The impetus for their studies is also the desire to predict air pollution dispersion. The papers published by LLL point out both the diversity which can exist in computer modeling and the marginal results that one often encounters when such numerical modeling is applied to wind prediction. In contrast with the LASL efforts, the model used at LLL (Ref. 7) employs variational techniques to obtain a time-independent, three-dimensional wind field model. Despite the sophisticated techniques involved, the comparison of numerical predictions with terrain measurements is not good. "Typically, 60% of the time the calculations of pollutant dispersion were within an order of magnitude (of terrain measurements)." The chief sources of error were found to be, in decreasing order of magnitude, (1) wind direction, (2) topography, (3) diffusion parameters, (4) source strength, and (5) wind speed. Probably because of its time-independent characteristics, the LLL program is shorter to run than corresponding LASL models; the LLL program takes 80% of the large core memory on a CDC – 7600 computer and uses 2 to 3 minutes of CPU time to generate approximately 30,000 grid points above the terrain. There is no documentation as to how many iterations (i.e., sweeps) are needed to achieve the steady-state conditions.

C. Work Performed at Colorado State University

Two university centers have also done important work regarding different aspects of wind modeling. Research done at the Colorado State University, Fort Collins (CSUFC), has centered on experimental wind tunnel modeling of atmospheric flows over various topographies. Specialized meteorological wind tunnels are used in these studies (Refs. 8 and 9). The wind tunnel at Ft. Collins is capable of simulating thermally stratified atmospheric boundary layers. This type of modeling is also being used in France, Australia, and New Zealand (Refs. 10-12). The foremost advantage to this approach is that it lowers costs if the terrain conditions can be reproduced accurately in a wind tunnel. Also, it becomes much cheaper to obtain pertinent data from wind tunnel measurements rather than from instrumentation erected and monitored on the actual terrain. The results published by CSUFC indicate that there exists enough similarity between wind tunnel patterns and actual wind fields to justify this approach (Ref. 13). It is still unknown how adequate this modeling can be in nonspecialized or nonmeteorological wind tunnels, and very little information about it has so far appeared in the literature (Ref. 14). The CSUFC interest in computer modeling seems to be centered on the analysis of atmospheric turbulence. Significantly, the results which were obtained from numerical models were then checked with wind tunnel measurements (Ref. 60).

D. Work at Other Universities and Centers

The other university center which has done long-range, sustained research on wind fields is Pennsylvania State University (PSU), where H. A. Panofsky and his co-workers

have conducted a number of investigations on the effect of terrain roughness on wind profiles (Refs. 15-19). Their data is probably the best record on how different types of terrain roughness such as forests, cropland, and desert can affect the wind profiles. How important this information would be in a numerical model of wind prediction, is not known at the present time. Work has also been done at Sandia Laboratories, Albuquerque, New Mexico on numerical wind modeling (Ref. 58), but the present status of this research is unknown.

The NASA-Lewis Research Center has been involved in designing and building their first 100-kW(e) wind propeller and working in cooperation with the Department of Energy as consultant in implementing similar or larger units at various other locations. However, relative to numerical modeling of wind velocities, it was not evident that NASA-Lewis was involved in detail.

E. JPL Activities

The wind prediction studies at JPL were aimed at determining the feasibility of using wind power as part of an energy system to supply the energy needs for the Deep Space Network at Goldstone, JPL in Pasadena, and Edwards sites. In 1974, a Wind Power Feasibility Study committee at JPL published a preliminary report (Ref. 59), which contained a section related to wind siting. The report limited itself to discussing observed wind speeds at different California sites. This data was based on a comprehensive report of observed wind speeds at 137 California sites (Ref. 42). In parallel to that effort, another program was undertaken to determine the effects of wind loading on the structure of the tracking antennas at Goldstone. For the latter program, continuous records of wind speed and direction were obtained at six Goldstone sites (including different heights at the same site) during the period from October 1974 through July 1976. These records form one of the most comprehensive wind logs available, despite considerable gaps, due mainly to equipment malfunctions. Besides the terrain measurements, a private consulting firm (MRI)[1] developed a computer model for wind prediction at the Goldstone site. The results of this program (Ref. 21) show it to be quite unreliable and very expensive at about $300/run. A possible cause for this poor performance is given as due to "lack of sufficient fineness in the rectangular grid representation of the terrain." At the present time, all of the data taken by the Goldstone wind energy measurement system has been processed and a report is expected to be released shortly.

[1] Meteorology Research, Inc., Box 637, 464 West Woodbury Road, Altadena, CA 91001.

F. Conclusions of Literature Search

From the literature search briefly described above, one can draw the following conclusions:

(1) No numerical model presently available can predict with sufficient accuracy wind speeds and directions over an arbitrary terrain.

(2) Building such a model is feasible; this has been shown by the computer models developed at LASL for pollutant dispersion.

(3) The results of any such model can be checked cheaply and accurately with the aid of wind tunnel experiments.

(4) JPL has one of the best data bases available to serve as an input for such a model. This is extremely important since a computer program is only as good as the inputs that the program starts with.

Building such a numerical model in a simple way would be of great help to the DSN in the decision of wind turbine locations to achieve a sizable reduction in the energy consumption and cost. The sections that follow address themselves to the "mechanics" that enter into writing such a new program. The fluid mechanics, mathematics, and computational bases for such a model are discussed in a tutorial manner starting from elementary notions and briefly touching some state-of-the-art techniques.

III. Mathematical and Physical Description of the Problem

The flow dynamics of many liquids and gases can be described by means of a mathematical expression known as the Navier-Stokes equation. This is a vector equation which relates the forces and accelerations acting on a fluid particle.

In cartesian coordinates, the x, y, and z components of this equation are respectively:

$$\frac{\partial V_x}{\partial t} + V_x \frac{\partial V_x}{\partial x} + V_y \frac{\partial V_x}{\partial y} + V_z \frac{\partial V_x}{\partial z} =$$

$$- \frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{\mu}{\rho} \left(\frac{\partial^2 V_x}{\partial x^2} + \frac{\partial^2 V_x}{\partial y^2} + \frac{\partial^2 V_x}{\partial z^2} \right) + F_x \tag{1}$$

$$\frac{\partial V_y}{\partial t} + V_x \frac{\partial V_y}{\partial x} + V_y \frac{\partial V_y}{\partial y} + V_z \frac{\partial V_y}{\partial z} =$$

$$-\frac{1}{\rho}\frac{\partial p}{\partial y} + \frac{\mu}{\rho}\left(\frac{\partial^2 V_y}{\partial x^2} + \frac{\partial^2 V_y}{\partial y^2} + \frac{\partial^2 V_y}{\partial z^2}\right) + F_y \tag{2}$$

$$\frac{\partial V_z}{\partial t} + V_x \frac{\partial V_z}{\partial x} + V_y \frac{\partial V_z}{\partial y} + V_z \frac{\partial V_z}{\partial z} =$$

$$-\frac{1}{\rho}\frac{\partial p}{\partial z} + \frac{\mu}{\rho}\left(\frac{\partial^2 V_z}{\partial x^2} + \frac{\partial^2 V_z}{\partial y^2} + \frac{\partial^2 V_z}{\partial z^2}\right) + F_z \tag{3}$$

The first term on the left hand side of these equations represents a temporal acceleration. The remaining terms on the left hand side represent accelerations due to the motion of the fluid (convective accelerations). The right hand side of the equations contains the force terms due respectively to pressure, viscous and bulk effects.

The three Navier-Stokes equations contain five unknowns: the velocity components V_x, V_y, and V_z, the pressure p and the density ρ. Even if the problem is such that ρ can be considered as a known constant, there will still be four unknowns. In order to have as many equations as there are unknowns, an additional equation is needed. The needed relation is the conservation of mass, known as the continuity equation. The continuity equation for an incompressible fluid is:

$$\frac{\partial V_x}{\partial x} + \frac{\partial V_y}{\partial y} + \frac{\partial V_z}{\partial z} = 0 \tag{4}$$

Hence, for the case where ρ is constant (incompressible flow), the continuity and the three components of the momentum equations (Navier-Stokes) will form a well-posed system of four equations and four unknowns.

An important characteristic of the Navier-Stokes equations is the nature of the convective acceleration terms. These accelerations consist of dependent variables (velocities) multiplied by their derivatives, for example V_x multiplied by $\partial V_x/\partial x$. Such terms are called "nonlinear", and the equations that contain these types of terms are called nonlinear equations. Nonlinear equations are very difficult to solve by analytic methods. A limited, if important number of solutions exist for simplified, linearized forms of the Navier-Stokes equations (Ref. 22). These simplified forms occur when the nature of the flow allows the investigator to neglect some of the terms in comparison with other terms. For example, for a flow which has the same axial velocity profile at all locations on the Z-axis, we can neglect terms that contain $\partial V_z/\partial z$.

It is only in the last decade that the advent of large computers has made possible the solution of the *full* Navier-Stokes equations by *numerical* methods. In considering such methods, one needs to know the answers to the following questions:

How are the equations approximated for numerical processing?

What are the numerical errors and instabilities that can arise?

What techniques are available to solve the flow problems of interest by fast, inexpensive, and accurate means?

Some answers to these questions are considered next.

IV. Numerical Procedures

A. Finite Differences and Finite Elements

There are two main numerical techniques available for the solution of fluid flow phenomena. They are the *finite-difference method* and the *finite-element method*. In the first, the differential equations to be solved are transformed into a set of *difference equations*. Difference equations are algebraic in nature. They consist of additions and subtractions of variables at discrete points, in place of the continuous derivatives that occur in differential equations.

In the second technique, *the finite element method*, variational calculus is used to solve the differential equations of the problem converting it to an integral form and following a minimization path. First, the region of interest is divided into discrete elements. The elements are assumed to be connected only at the common nodes of their boundaries, (for example, nodes a, b, c, in Fig. 1). The finite-element method searches for the values of the dependent variables at all these common nodes within the domain. The variational techniques which are used (for example, the Bubnov-Galerkin method) consist of setting up integrals of the differential equations and then minimizing the values of these integrals at each of the nodes in the domain.

There is no clear-cut choice as to which of these two methods is better for the solutions of fluid mechanics problems. Finite differences are easier to set up than finite

elements. In addition, there is a large body of experience dealing with finite difference techniques as applied to numerical fluid dynamics (Ref. 23). This experience has to do with questions of numerical stability and convergence and is very important when writing new numerical schemes. The main disadvantage of the finite-difference method is that it does not lend itself very easily to incorporating irregular physical boundaries.

On the other hand, finite element schemes, originally devised for structural and elasticity problems, have only recently been applied to fluid mechanics problems. Their great advantage is the ease with which the solution domain can be discretized. Therefore, irregular physical boundaries are easily described by this method. However, finite-elements methods suffer from difficulties in dealing with nonlinear terms of the Navier-Stokes equations and with incorporating the incompressibility conditions. While new methods are being devised to overcome these difficulties (Refs. 24, 25), finite elements methods also seem to require slightly more computing time than finite difference methods (Ref. 26).

It can be shown (Ref. 27) that finite differences are written for points in space and finite elements are spatial integrators of point formulas. However, no comparisons have been published regarding the efficiency and cost of these two methods when applied to the same problem. Mainly because of the difficulties still remaining with the finite element methods, we will concentrate our attention to the solution of the Navier-Stokes and continuity equations by means of finite-difference techniques.

B. Derivatives and Computational Meshes

There are a number of ways by which derivatives can be approximated by finite differences. Suppose, for example, that one is interested in computing the changes in the x-component of the velocity vector with respect to the position on the x-axis, that is $\partial V_x/\partial x$. Consider a computational mesh, with equal mesh intervals Δh, in the x-direction, as shown in Fig. 2. Denote by $(n-1)$, (n) and $(n+1)$ a series of three adjacent points at the intersections of the grid lines. These points are on a grid line parallel to the x-axis. In general, the velocity component V_x has different values at each of these points; these values are denoted respectively by V_{n-i}, V_n, and V_{n+i}. The change in V_x with respect to x, *evaluated at the point n*, can be approximated by the differences between the velocity values at these points. For example, for the *forward-difference scheme*:

$$\frac{\partial V_x}{\partial x} \cong \frac{V_{n+1} - V_n}{x_{n+1} - x_n} = \frac{V_{n+1} - V_n}{\Delta h} \tag{5}$$

For the *backward-difference scheme*:

$$\frac{\partial V_x}{\partial x} \cong \frac{V_n - V_{n-1}}{x_n - x_{n-1}} = \frac{V_n - V_{n-1}}{\Delta h} \tag{6}$$

Finally, for the *central-difference scheme*, we can take differences relating points on either side of n and obtain:

$$\frac{\partial V_x}{\partial x} \cong \frac{V_{n+1} - V_{n-1}}{x_{n+1} - x_{n-1}} = \frac{V_{n+1} - V_{n-1}}{2\Delta h} \tag{7}$$

Because all difference formulas are only approximations to derivatives, they will contain some errors when compared with the actual values of the derivatives. The magnitude of the errors depends on the finite difference form used. It can be shown mathematically (Ref. 28) that of all three schemes presented above, the central-differences gives the most accurate result.

In general, velocities and pressures do not vary in one direction only. In rectangular coordinates, two- or three-dimensional computational cells, as those shown in Figs. 3a and 3b, are often used.

For two- or three-dimensional flows, one has to decide how to assign the variables *within* each cell. For the two-dimensional case, the most popular schemes are illustrated in Figs. 4a and 4b.

In Fig. 4a, averages at the *center* of the cell are used to determine fluxes through the cell's edges. The staggered mesh arrangement shown in Fig. 4b is more advantageous for incompressible flows (Ref. 29). Here, the horizontal velocity component V_x is centered on the right and left faces of the cell, while the vertical component V_y is centered on the bottom and top. The pressure and density are cell-center averaged.

In addition to assigning the variables within each cell, one also has to "optimize" the computational grid size. An optimal grid size is one which yields good resolution for the fluid flow phenomena. It is natural to expect that the smaller the grid size, the better the resolution. However, a small grid size also implies a greater number of cells covering the domain of interest. With a larger number of cells, both computational time and cost will increase. Therefore, a good compromise is to use a variable mesh size having small cells in regions of rapid flow changes and larger cells in regions where the flow does not change drastically (Refs. 3 and 36).

In the next section, it will be shown that the selection of mesh size and the way the variables are assigned within each cell are part of the computational stability problem. Meaningful solutions of the Navier-Stokes equations can be obtained only when written in a certain form and when the finite differences are forced to take into account the direction of the flow velocities.

C. Problems of Stability and Convergence

When finite-differences are substituted for the derivatives, it may happen that the numerical solution behaves very differently compared to the solution of the corresponding differential equation. The main causes for such discrepancies are instability and poor convergence.

Since any computer memory is limited, the numbers that are stored and worked with in computers can be represented only by a finite number of digits. Numbers having more digits are "rounded-off" according to algorithms designed into the machine. Any such rounding-off will contribute to errors in the final result. Therefore, it is possible that round-off errors will propagate through a computer program and, as a result, small changes in the initial data will be translated into very large changes in the output. Numerical schemes that suffer from such defects are said to be unstable. The growth of errors due to the presence of possible extraneous solutions to the difference equations, can also cause instability (Ref. 61).

Another possible type of error has to do with the convergence of the algorithm used. In general, finite differences are only approximations to the continuous derivatives that they replace, and therefore, the numerical solution will always be in error compared to the solution of the original differential equation. Mathematically, one defines a *convergent* finite difference scheme as one in which the finite differences solution approaches the continuum differential equation solution, as the mesh size approaches zero (Refs. 23 and 35).

The definitions of stability and convergence can provide only rough guidelines for practical numerical calculations. In particular, stability analysis is still in its infancy. The oldest method for testing the stability of a numerical scheme is due to von Neumann (Ref. 31) and is based on expressing the dependent variables in terms of Fourier series. When the Fourier components are substituted into the finite-difference equation, the decay or growth of each mode in these components will show whether the scheme is stable or unstable, respectively. While this technique works fairly well for linear equations with constant coefficients, it is inaccurate when applied to nonlinear equations such as the Navier-Stokes equations. Hirt (Ref. 32) has introduced another method, based on expanding each term of the finite difference equation into Taylor series and analyzing the higher order terms. Although Hirt's method can be applied to nonlinear equations with variable coefficients, it is uncertain yet whether it can be applied to some of the common complex equations.

It will be shown later that computational stability can be improved by a judicious choice of finite-difference expressions. In particular, a certain scheme of differentiation, called the "upwind" or "donor-cell" method, has been found to greatly improve the stability of the numerical solutions of the Navier-Stokes equations. However, before this method is described further, convergence must be looked at in more detail.

As mentioned earlier, convergence is related to the accuracy of the numerical solution, and it is an indication of how close the numerical solution approximates the solution of the continuous differential equation. The laws of fluid mechanics, as given by Eqs. (1), (2), (3), and (4) can be written in different forms, to which correspond different numerical schemes. Some of these schemes have better convergence than others. In particular, when the momentum equations are written in what is called a "conservative form," it has been argued (Refs. 29, 33 and 34), that the finite difference equations corresponding to this form give more accurate results than when the equations are in a "non-conservative" form.

Although there is no rigorous proof for this statement, and some exceptions have been found (Ref. 23), in general, it is advisable to write the momentum equations in the conservative form (Ref. 35).

D. Conservation: "Upwind-Differencing"

An equation is said to be in "a conservative form" when it can be written as a sum of a time derivative and a spatial flux. For example, the *one-dimensional* momentum equation can be written in conservative form as

$$\frac{\partial(\rho V_x)}{\partial t} + \frac{\partial}{\partial x}\left(\rho V_x^2 + \rho\right) = 0 \tag{8}$$

The corresponding "non-conservative" form of the momentum equation is:

$$\frac{\partial V_x}{\partial t} + V_x \frac{\partial V_x}{\partial x} + \frac{1}{\rho}\frac{\partial p}{\partial x} = 0 \tag{9}$$

Note that Eq. (8) can be obtained from Eq. (9) by adding to the latter the continuity equation multiplied by V_x. The corresponding finite-difference equations for Eqs. (8) and (9) are, respectively,

$$\frac{(\rho V_x)_j^{t+\Delta t} - (\rho V_x)_j^t}{\Delta t} - \frac{1}{\Delta x}\left[(\rho V_x^2 + p)_{j+1} - (\rho V_x^2 + p)_{j-1}\right] = 0 \tag{10}$$

and

$$\frac{(V_x)_j^{t+\Delta t} - (V_x)_j^t}{\Delta t} + (V_x)_j^t \frac{(V_x)_{j+1}^t - (V_x)_{j-1}^t}{\Delta x} + \frac{1}{\rho_j}(P_{j+1} - P_{j-1}) = 0 \tag{11}$$

In the above equations, the superscripts refer to the time step and the subscripts refer to the computational cell (see Fig. 5).

Note than in Eq. (10), the change of momentum with respect to time is given by the difference of cell edge fluxes. In the second equation, neither the velocity nor the pressure are written in a conservative form, because the cell-edge difference fluxes, that is $(p_{j+1} - p_{j-1})$ and $(V_x)_{j+1}^t - (V_x)_{j-1}^t$, are multiplied by $(1/\rho_j)$ and $((V_x)_j^t)$, respectively.

In particular, for the velocity term, Fig. 5 and Eq. (11) indicate that the flux out of the right side of cell j and which enters cell $j+1$, is $((V_x)_j(V_x)_{j+1/2})$. However, the flux *into* cell $(j+1)$, *from* cell j, is $((V_x)_{j+1}(V_x)_{j+1/2})$. Therefore, at the interface between two cells, the outgoing flux is different than the incoming one and it is for this reason that the numerical scheme in Eq. (11) is called "non-conservative."

It has been shown that the finite difference form of the Navier-Stokes equations is frequently unstable, even when the equations are written in conservative forms (Refs. 4, 36-38). However, these equations can be made stable, by replacing the central differences scheme by the "upwind" or "donor-cell" differencing scheme (Ref. 39). In this method, the boundary values for any dependent variable are a function of the direction of the flow velocity. For example, labeling this dependent variable by Q, the upwind (donor-cell) scheme gives:

$$Q_{j+1/2}^t = \begin{cases} Q_j^t & \text{if } (V_x)_{j+1/2}^t \geq 0 \quad (12) \\ Q_{j+1}^t & \text{if } (V_x)_{j+1/2}^t < 0 \quad (13) \end{cases}$$

This is illustrated in Figs. 6a and 6b.

V. Simplification of the Governing Equations for Wind Modeling

The Navier-Stokes equation, as written in Section III, is very complicated and its numerical solution would require a great deal of computer time and money. Therefore, in constructing a numerical scheme, it is useful to see if the equations can be simplified in any way. For example, in the problem of wind prediction, by using order-of-magnitude analysis one can use wind speed and direction measurements to check if any of the force and/or acceleration terms are significantly smaller than the other terms. As indicated in Section I, such measurements do exist for the Goldstone area and this data can be used to answer the following questions: (1) Is it necessary to keep the temporal acceleration (transient term) in the Navier-Stokes equation? (2) Are all the convective acceleration terms significant, or can the flow in one direction be neglected in comparison with the other two? (3) Can viscous effects be neglected and the problem be treated as inviscid flow? (4) How can thermal heating and stratification be entered into the equations? (5) Are the flow conditions laminar or turbulent? This section attempts to answer these questions, based on the Goldstone data and on other measurements.

A. Variations in Wind Speed and Direction at Goldstone

The data taken at Goldstone were processed by calculating hourly averages of wind speed and direction. Changes in this average are illustrated in Fig. 7 for an arbitrarily selected week in October 1974. A trend of stronger afternoon winds may be discerned, but this trend is not reproducible enough to be incorporated in any numerical model. It seems that on a diurnal basis there is no steady state for wind velocity, and the best approach is to accept an hourly quasi steady state, with values given by the averages processed from the Goldstone data. Although this means that such a model will have to be run for every hour, the model itself will be much simpler to run than a transient model and thus the cost per run will be lower. The usefulness of this approach has been recognized by other investigators who have built models based partially on this assumption (Ref. 40). An additional advantage of this quasi-steady state approach is that temporary variations in turbulent eddy structures will be averaged out; this will also contribute in decreasing the complexity of the model. (A note on turbulence and its effects appears in Section C.)

B. Convective Accelerations in Wind Fields

In the rectangular coordinate system, there are three mutually perpendicular velocity components: V_x, V_y, and V_z. Spatial changes in these components (e.g., $\partial V_x/\partial x$, $\partial V_y/\partial x$, $\partial V_z/\partial y$) multiplied by the velocities themselves make up the

convective accelerations. The Goldstone data indicate that wind magnitudes do change from location to location, but that velocity changes with height area are, in general, much smaller than those in the horizontal plane (see Fig. 8). Such an observation is also supported by measurements taken at the Savannah River Plant, Georgia (see Fig. 9)(Ref. 41). Note, however, that this data represents speed, a scalar, and not velocity, a vector. Indeed, Fig. 10 indicates that the wind direction changes with height, this change in direction remaining nearly constant with time. Other data, however, indicate a logarithmic distribution of the horizontal velocity components (Refs. 8, 42, 43 and 57).

$$\overline{U}_Z = \frac{U^*}{K} ln\left(\frac{Z}{Z_0}\right) \tag{14}$$

In this formula, $\overline{U}_Z$ is the mean speed (m/s) at height Z above the ground, U^* is the friction velocity[2] (m/s), K is von Karman's constant ($\simeq$0.04), and Z_0 is the roughness length (m). This last quantity is terrain dependent and is related to the height of the roughness elements in a given terrain, such as trees, buildings, crops, etc. Examples of Z_0 are:

Terrain	Roughness length Z_0, m
Sand, desert	0.0003
Grass	0.003 - 0.01
Agricultural crops	0.04 - 0.20
Forests	1.0 - 6.0
City	1.5
Snow	0.00005 - 0001

It is possible that these conflicting data sets result from different wind velocity profiles occurring at different wind speeds, and that these changes may also depend on the surface roughness. In the model proposed, the acceleration normal to the terrain will be taken into account so as to allow the model to exhibit terrain sheltering and some aspects of flow separation (Ref. 40).

C. Viscous Effects, Turbulence

The problems of viscous effects are tied to the velocity profiles. If normal accelerations are to be allowed as indicated in the previous paragraph, then a full three-dimensional model is needed. Such a model must take into effect the influence of viscosity, i.e., the boundary layer that is developed between the main air flow and the ground. Tied to the nature of boundary layers is the problem of turbulence, one of the most complex in fluid dynamics. The modeling of turbulence in any numerical code is not only difficult, but of uncertain accuracy also. Because of this, at least in the preliminary program, turbulence effects will not be taken into account.

[2] U^* is a measure of surface shear stress.

D. Thermal Heating

It was mentioned in Section II that body forces are important in the setup of the Navier-Stokes equations. In the problem of air flow, body forces due to gravitational effects must be coupled with buoyancy effects. The latter are due to thermal heating or, more precisely, to differences in the temperature between the air close to the ground (warmer air) and air higher up (cooler air). These differences in temperature cause differences in density and thus buoyancy forces. The Boussinesq parameter accounts for these forces and also incorporates the effect of gravity. This parameter can be written in either of the two forms:

$$\left(\frac{\rho}{\rho_0} - 1\right) \mathbf{g} = -\beta(T - T_0)\mathbf{g} \tag{15}$$

In these expressions, g is the gravitational acceleration, ρ_0 and T_0 are reference density and temperature, respectively, and β is the volumetric expansion coefficient.

Note that the second form involves temperatures, which must be considered as unknowns for the momentum and continuity system of equations; this implies that an additional equation, the *energy equation*, must be added to the system.

It is not immediately apparent how important the effect of density stratification is on the prediction of wind speeds. Most wind model studies, (e.g., Ref. 44) do not compare predicted results that include stratification with results that do not. As a compromise between accuracy and computational complexity, the effect of density stratification will be retained, but instead of adding the energy equation to our previous system, the new model proposed will incorporate a subroutine giving the explicit dependence between density and temperature. The advantages of such an approach are also discussed in Ref. 45.

E. Simplified System of Equations

As a result of the above discussions, the system to be solved is composed of the following equations:

Continuity:

$$\frac{\partial V_x}{\partial x} + \frac{\partial V_y}{\partial y} + \frac{\partial V_z}{\partial z} = 0 \tag{16}$$

Momentum:

$$V_x \frac{\partial V_x}{\partial x} + V_y \frac{\partial V_x}{\partial y} + V_z \frac{\partial V_x}{\partial z} =$$

$$-\frac{1}{\rho}\frac{\partial p}{\partial x} + \frac{\mu}{\rho}\left(\frac{\partial^2 V_x}{\partial x^2} + \frac{\partial^2 V_x}{\partial y^2} + \frac{\partial^2 V_x}{\partial z^2}\right) \quad (17)$$

$$V_x \frac{\partial V_y}{\partial x} + V_y \frac{\partial V_y}{\partial y} + V_z \frac{\partial V_y}{\partial z} =$$

$$-\frac{1}{\rho}\frac{\partial p}{\partial y} + \frac{\mu}{\rho}\left(\frac{\partial^2 V_y}{\partial x^2} + \frac{\partial^2 V_y}{\partial y^2} + \frac{\partial^2 V_y}{\partial z^2}\right) \quad (18)$$

$$V_x \frac{\partial V_z}{\partial x} + V_y \frac{\partial V_z}{\partial y} + V_z \frac{\partial V_z}{\partial z} =$$

$$+\frac{1}{\rho}\frac{\partial p}{\partial z} + \frac{\mu}{\rho}\left(\frac{\partial^2 V_z}{\partial x^2} + \frac{\partial^2 V_z}{\partial y^2} + \frac{\partial^2 V_z}{\partial z^2}\right) + g\left(\frac{\rho}{\rho_0} - 1\right) \quad (19)$$

This system contains four equations and four unknowns: V_x, V_y, V_z, p. As it stands, the system also needs explicit relationships between the density and temperature:

$$\rho = \rho(T) \quad (20)$$

and the viscosity and temperature

$$\mu = \mu(T) \quad (21)$$

A final point worth mentioning is the problem of compressibility. In the equations presented above, air is treated as an incompressible fluid. Airflows at moderate temperatures, say between 50 and 100°F, and at speeds less than about 100 m/s (225 mph), can be considered incompressible (Ref. 46). Since these limits are well beyond the situations expected in this study of winds, the numerical analysis will be based on an incompressible flow.

VI. Steady-State Models

The differential equations given in the previous section describe steady, fully developed, three-dimensional flows. Effective numerical procedures for the solution of these equations have been worked out by a group of investigators led by D. B. Spalding (Refs. 34, 47 and 48).

The main variables in these procedures are the velocities and the pressures. The domain of interest is overlaid by a rectangular mesh which can contain cells of different size. The differential equations are then approximated by finite differences applied to this mesh. Figure 11 illustrates the placement of the variables within the computation grid.

Note that in this scheme, the variables are staggered, meaning that the pressure and velocities are stored at different locations; the pressure is stored at the intersection of the grid lines, e.g., at P, and the velocities at the locations indicated by the arrows, i.e., midway between the grid lines. (In this figure, the velocity components for point P are V_x and V_y).

The reason for using a "staggered-grid" arrangement is that momentum fluxes can be easily computed from the *integration* of the velocity components over the areas A_x and A_y, and such integrations form the basis of these methods (Ref. 62).

From the integration of the momentum differential equations, and by using a combination of central and upwind difference schemes, algebraic equations are obtained involving the velocity components and/or any scalar quantity that might be needed such as effluent concentration, radiation flux, etc. Together with the finite difference form of the continuity relation, this set of equations is solved by a semi-implicit[3] method.

The steps in this method are first to establish guessed pressures (either as pure guesses or as values from a previous calculation sweep), and then to obtain a field of intermediate velocities. In general, these intermediate velocities do not satisfy the continuity equation and a pressure correction term must then be introduced. The process of finding this pressure correction field reduces to solving a Poisson equation, a problem for which there exist many rapid and economical methods of solution (Ref. 49).

The algorithm used in these papers has been applied to problems such as that of laminar flow over a slab-sided "build-

[3]The terms explicit and implicit appear quite often in the description of solutions for partial differential equations. Explicit methods are those in which the unknown quantity, say a velocity at a time t, V, is determined in terms of previously found quantities, say V^{t-1}, V^{t-2}, . . . , V^{t-n}. In the implicit methods, *two or more* values of the unknown quantity are specified in terms of known values on a previous data line. For example, the values of say the velocity at time t *and* at different locations can be described as a function of the velocities at these locations at a time $(t-1)$. Implicit methods are often used in time-dependent heat conduction problems.

ing" (Ref. 48), flow with heat transfer in three-dimensional ducts (Ref. 50), and boundary layer flows (Ref. 5). The results are quite good and the methods appear economical, although no figures are given.

There exist some restrictions imposed on these steady programs. These limitations are of three kinds:

(1) All the papers cited above deal with problems involving low Reynolds number laminar flows. At higher Reynolds numbers, which might be expected in wind fields, *turbulence may become a factor.* In this case, one would have to incorporate a turbulence model such as those outlined by Gawain and Pritchett (Ref. 52) or by Deardorff (Ref. 53).

(2) The boundaries of the domain of integration are rigid and impermeable to fluid flow. This simplifies the boundary conditions. In a real situation, however, topographical variations will affect the wind field at the boundaries.

(3) Finally, all the problems under consideration have dealt with regular geometrics: tubes, rectangular blocks, etc. In the wind flow problem, the topography is highly irregular and storing the necessary terrain description can be a significant problem.

None of the difficulties described above represent insurmountable obstacles. As mentioned above, numerical schemes that describe turbulence already exist in the literature. The problem of irregular topography has been attacked by Viecelli (Ref. 54) and also by Mason and Sykes (Ref. 55). Extrapolation formulas can be used to describe the boundary flows; an example of this approach appears in Clark (Ref. 56). It seems possible, therefore, to build on these works and write an accurate wind-field prediction model.

VII. Conclusions and Plans for Future Work

The present report has surveyed various numerical methods which can be used to predict the wind field over an arbitrary terrain, together with an overview of modeling activities by various centers. Several conclusions can be drawn from this review.

Writing such a numerical code is feasible. Numerous papers have been cited which successfully predict the dispersal of pollutants and the velocity vectors over slablike, cuboidal and bell-shaped obstacles. There is no intrinsic and impenetrable difficulty in predicting the velocities over an irregular, three-dimensional terrain such as one would encounter in real life. No claim is made that the problem is simple and that all one has to do is to slap together previously worked out techniques; however, writing such a numerical code is perfectly feasible.

Due to the nature of the problem which requires the manipulation of large amounts of data, any wind prediction numerical model must be economical to run. This economy can be gained by various means – the most obvious one being good programming. Beyond this purely technical aspect, however, the physical data of the problem can also be used to reduce the computational load. For example, by assuming an incompressible and steady-state flow, the governing equations can be simplified considerably and the computational complexity is reduced. The only limit in this simplification process is that the resulting physical model must give a wind field prediction reasonably close to terrain measurements.

A numerical wind model along the lines described above is in the process of being written in the DSN Advanced Engineering and Energy Conservation Group. This model will take advantage of the existing Goldstone Wind Data Base. The results of the computer program will be checked both against terrain measurements and the wind tunnel simulations.

References

1. Harlow, F. H., and Welch, J. E., "Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface," *Phys. Fluids*, Vol. 8, No. 12, pp. 2182-2189, 1965.

2. Harlow, F. H., "The Particle-in-Cell Computing Method for Fluid Dynamics," *Methods in Computational Physics*, Vol. 3, p. 318 ff, 1964.

3. Hirt, C. W., Ramshaw, J. D., and Stein, L. R., "Numerical Simulation of Three Dimensional Flow Past Bluff Bodies," *Comp. Methods Appl. Mech. Engr.*, 14: 93-124, (1978).

4. Hirt, C. W., and Cook, J. L., "Calculating Three-Dimensional Flows Around Structures and over Rough Terrain," *J. Comput. Phys.*, 10: 324-340, 1972.

5. Nichols, B. D., and Hirt, C. W., "Transient Three-Dimensional Fluid Flow in the Vicinity of Large Structures," in Lectures Notes in Physics V. 19, Proc. of the Third Int. Conf. in Numerical Methods in Fluid Mechanics, Springer-Verlag (1972).

6. Nakayama, P. I., and Romero, N. C., "Numerical Method for Almost 3D Incompressible Fluid Flow and a Simple Internal Obstacle Treatment," *J. Comput. Phys.*, 8: 230-240, 1971.

7. Sherman, C. S., "MATHEW – A Mass Consistent Wind Field Model," Lawrence Livermore Laboratory, Report UCRL-52479, May 1978.

8. Cermak, J. E., "Laboratory Simulation of the Atmospheric Boundary Layer," AIAA Paper No. 70-751, 1970.

9. Hsi, G., Binder, G. J., and Cermak, J. E., "Topographic Influences on Wind Near Green River, Utah," Colorado State Univ., Fort Collins, Fluid Mechanics Program, Report No. CER-67-68GH-GJB-JEC54, 1968.

10. Mery, P., "Scale Reproduction of Atmospheric Boundary Layer Diffusion," La Houille Blanche, No. 3, 1969.

11. Mulhearn, P. J., "Turbulent Flow over a Very Rough Surface," 6th Tech. Conf. of Australian Hydraulics and Fluid Mechanics, Adelaide, Australia, 1977.

12. Raine, J. K., "Simulation of a Neutrally Stable Rural Atmospheric Boundary Layer in a Wind Tunnel," 5th Tech. Conf. of Australian Hydraulics and Fluid Mechanics, Christchurch, New Zealand, 1974.

13. Cermak, J. E., and Peterka, J., "Simulation of Wind Fields over Point Arguello, Calif., by Wind-Tunnel Flow over a Topographic Model," Colorado State Univ., Fort Collins, Rept. CER65 JEC-JAP 64, Nov. 1966.

14. Boutwell, F. K., and Porter, R. E., "Industrial Applications of Wind Tunnel Investigations" AGARD Rept. 314 Oct. 1960.

15. Blackadar, A. K., and Panofsky, H. A., "Effects of a Change of Terrain on Wind Profiles," Final Report, July 1, 1965 – August 15, 1968, Pennsylvania State Univ., Dept. of Meteorology, Univ. Park, Penn. 1970.

16. Panofsky, H. A., and Townsend, A. A., "Change of Terrain Roughness and the Wind Profile," Quart. J. of the Royal Meteorological Society, 90 (384): 147-155, April 1964.

17. Panofsky, H. A., and Petersen, E. L., "Wind Profiles and Change of Terrain Roughness at Riso," Quart. J. of the Royal Meteor. Soc. 98 (418): 845-854, Oct. 1972.

18. Peterson, E. W., "On the Relation Between the Shear Stress and the Velocity Profile After a Change in Surface Roughness," *J. Atmos Sci.*, 26 (4): 773-774, July 1969.

19. Blackadar, A. K., Glass, P. E., and Panofsky, H. A., "Effect of Stability on the Wind Profile Under Conditions of Inhomogeneous Terrain Roughness," Symposium on the Theory and Measurement of Atmospheric Turbulence and Diffusion in the Planetary Boundary Layer, Albuquerque, N.M., Dec. 5-7, 1967.

20. Wolte, C. G., Rhody, T. R., Goodridge, J. D., and Powers, J. B., "Office Report on Wind in California," State of California Dept. of Water Resources, Division of Resources Planning, Climatology Group, August 1960.

21. Levy, R., and McGinness, H., *Wind Power Prediction Models*, TM 33-802, Jet Propulsion Laboratory, Pasadena, Calif., Nov. 1976.

22. Berker, R., "Integration Des Equations Du Movement D'un Fluide Visqueux Incompressible," *Handbuch der Physik*, Band VIII/2, Springer, Berlin, 1963.

23. Roache, P. J., *Computational Fluid Mechanics*, Hermosa Publishers, Albuquerque, N.M., 1972.

24. Hughes, T. J. R., Liv, W. K., and Brooks, A., "Finite Element Analysis of Incompressible Viscous Flows by the Penalty Function Formulation," *J. Comput. Physics*, 30: 1-60, 1979.

25. Schneider, G. E., Raithby, G. D., and Yovanovich, M. M., "Finite Element Solutions Procedure for Solving the Incompressible Navier-Stokes Equations Using Equal Order Variable Interpolation," *Numerical Heat Transfer*, Vol. 1, pp. 433-451, 1978.

26. Engleman, M., "A Finite Element for the Numerical Solution of Viscous Incompressible Flows," *J. Comput. Phys.*, 30: 181-201, 1979.

27 Gray, W. G., and Pinder, G. F., "On the Relationship Between the Finite-Element and Finite Difference Methods," *Int. J. for Numerical Methods in Engineering*, 10: 893-923, 1976.

28. Forsythe, G. E., and Wasow, W. R., *Finite-Difference Methods for Partial Differential Equations*, J. Wiley and Sons, N.Y. (1960).

29. Harlow, F. W., and Amsden, A. A., "Fluid Dynamics," Los Alamos Scientific Laboratory Report, LA-4700, Los Alamos, N.M., June 1971.

30. Kalnay de Rivas, E., "On the Use of Nonuniform Grids in Finite-Difference Equations," *J. Comput. Phys.*, 10: 202-210, 1972.

31. O'Brien, G. G., Hyman, M. A., and Kaplan, S., "A Study of the Numerical Solution of Partial Differential Equations," *J. Math Phys.*, 29: 233-251, 1950.

32. Hirt, C. W., "Heuristic Stability Theory for Finite Difference Equations," *J. Comput. Phys.*, 2: 339-355, (1968).

33. Cheng, S. I., "A Critical Review of Numerical Solution of Navier-Stokes Equations," in Lecture Notes in Physics, V. 41, "Progress in Numerical Fluid Dynamics," ed. H. J. Wirz, p. 146, 1974.

34. Runchal, R. K., "Convergence and Accuracy of Three Finite Difference Schemes for a Two-Dimensional Conduction and Convection Problem," *Int. J. Num. Methods Engr.*, 4: 541-550, 1972.

35. Ames, W. F., *Numerical Methods for Partial Differential Equations*, 2nd. ed., Academic Press, N.Y. (1977).

36. Hirt, C. W., Nichols, B. D., and Romero, N. C., "SOLA – A Numerical Solution Algorithm for Transient Fluid Flows," Los Alamos Scientific Laboratory Report, LA-5852, Los Alamos, N.M., April 1975.

37. Daly, B. J., and Pracht, W. E., "Numerical Study of Density-Current Surges," *Phys. Fluids*, 11(1): 15-30, (1968).

38. Roache, P. J., "On Artificial Viscosity," *J. Comput. Phys.*, 10:169-184, 1972.

39. Gentry, R. A., and Martin, R. E., "An Eulerian Differencing Method for Unsteady Compressible Flow Problems," *J. Comput. Phys.*, 1:87-118, 1966.

40. Ball, J. A., and Johnson, S. A., "Physically Based High Resolution Surface Wind and Temperature Analysis for EPAMS," Mission Research Corporation, Santa Barbara, Calif., Rept. MRC-R-7731-1-278.

41. Sherman, C. S., "MATHEW: A Mass-Consistent Wind Field Model," Lawrence Livermore Laboratory Report, UCRL – 52479, May 1978.

42. Oke, T. R., *Boundary Layer Climates*, Methuen & Co., Ltd., London, 1978.

43. Bietry, J., Sacre, C., and Simiu, E., "Mean Wind Profiles and Change of Terrain Roughness," Proceedings of the ASCE, J of the Struct. Division, V. 104, No. ST 10, Oct. 1978, pp. 1585-1593.

44. Agopian, K. G., and Crow, S. C., "The Effect of Atmospheric Density Stratification on Wind Turbine Siting," DOE Report RLO-2444-78/1, Jan. 1978.

45. Ball, J. A., "A Concept for a High-Resolution Toposcale Wind Model to Estimate Surface Wind in Complex Terrain," Mission Research Corporation (MRC) March 15, 1979, Santa Barbara, Calif.

46. Mironer, A., *Engineering Fluid Mechanics*, McGraw-Hill Book Co., N.Y., 1979.

47. Spalding, D. B., "A Novel Finite Difference Formulation for Differential Expressions Involving Both First and Second Derivatives," *Int. J. Num Meth. Engr.*, 4: 551-559, 1972.

48. Caretto, L. S., Gosman, A. D., Patankar, S. V., and Spalding, D. B. "Two Calculation Procedures for Steady, Three-Dimensional Flows with Recirculation," in Proceedings of the 3rd Int. Conf. on Numerical Methods in Fluid Mechanics, Vol. II, 1972, Springer-Verlag, New York.

49. Burridge, D. M., and Temperton, C., "A Fast Poisson – Solver for Large Grids," *J. Comput. Phys.* 30: 145-148, 1979.

50. Pratap, V. S., and Spalding, D. B., "Fluid Flow and Heat Transfer in Three Dimensional Duct Flows," *Int. J. Heat Mass Transfer*, 19:1183-1188, 1976.

51. Caretto, L. S., Curr, R. M., and Spalding, D. B., "Two Numerical Methods for Three-Dimensional Boundary Layers," *Comput. Meth. in Appl. Mech. Engr.*, I: 39-57, 1972.

52. Gawain, T. H., and Prichett, J. W., "A Unified Heuristic Model of Fluid Turbulence," *J. Comput. Phys., 5*, 1970.

53. Deardorff, J. W., "On the Magnitude of The Subgrid Scale Eddy Coefficient," *J. Comput. Phys.*, 7: 120-133, 1971.

54. Viecelli, J. A., "A Computing Method for Incompressible Flows Bounded by Moving Walls," *J. Comput. Phys.*, 8: 119-143, 1971.

55. Mason, P. J., and Sykes, R. I., "Three Dimensional Numerical Integrations of the Navier-Stokes Equations for Flow Over Surface-Mounted Obstacles," *J. Fluid Mech.*, 91 (3): 433-450, 1979.

56. Clark, T. L., "A Three Dimensional Small Scale Dynamic Model Using a Terrain Following Co-ordinate Transformation," *J. Comput. Phys.*, 24: 186 ff, 1977.

57. Simiu, E., "Logarithmic Profiles and Design Wind Speeds," *J. of Engr. Mech.*, Division, ASCE Vol. 102, No. ST. 4 Proc. Paper 10100, Oct. 1973, pp. 1073-1083.

58. Gartling, D. K., "NACHOS: A Finite Element Computer Program for Incompressible Flow Problems. Part I. Theoretical Background, SANDIA Laboratories, Report SAND-77-1333, NTIS ACCESS, N78-33395, 1977.

59. Clements, P., McGinness, H., and Lutes, G., "Wind Study," Appendix C to the JPL Wind Power Feasibility Study Committee, August 1974 (an internal document).

60. Huang, C. H., and Nickerson, E. C., "Numerical Simulation of Wind, Temperature, Shear Stress and Turbulent Energy Over Nonhomogeneous Terrain," Colorado State Univ., Fort Collins, Rept. CER71-72CH-ECN 23 March 1972.

61. Hornbeck, R. W., "Numerical Marching Techniques for Fluid Flow with Heat Transfer," NASA SP-297, 1973.

62. Gosman, A. D., Pun, W. M., Runchal, A. K., Spalding, D. B., and Wolfshtein, M., *Heat and Mass Transfer in Recirculating Flows*, Academic Press, 1969.

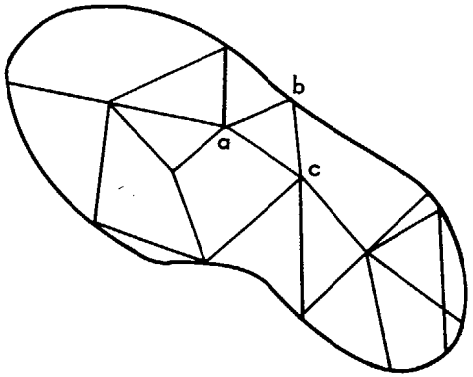

Fig. 1. Finite elements

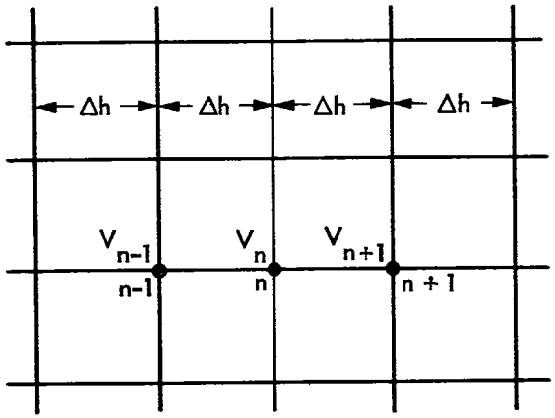

Fig. 2. Computational mesh for one-dimensional velocity derivative

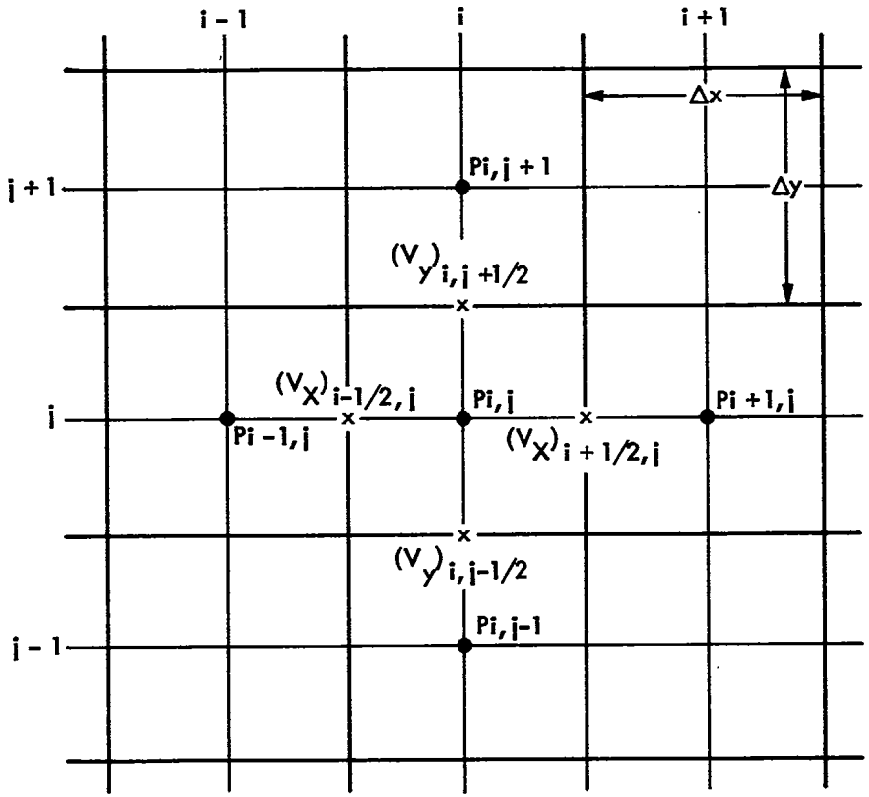

Fig. 3a. Two-dimensional case

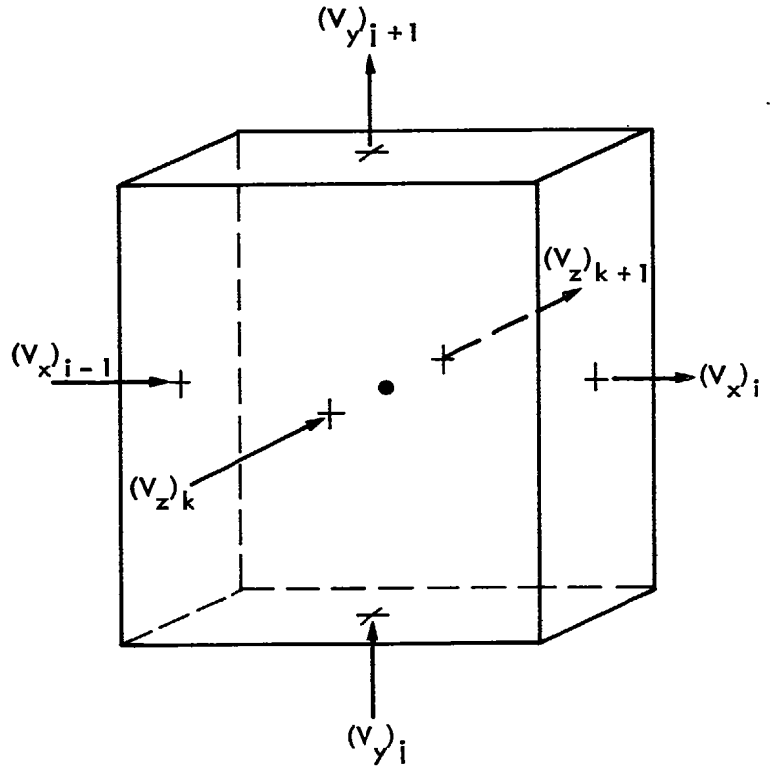

Fig. 3b. Three-dimensional case

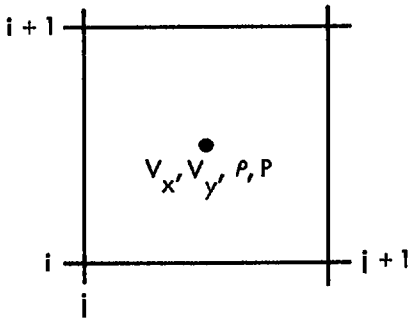

Fig. 4a. Cell centered variables

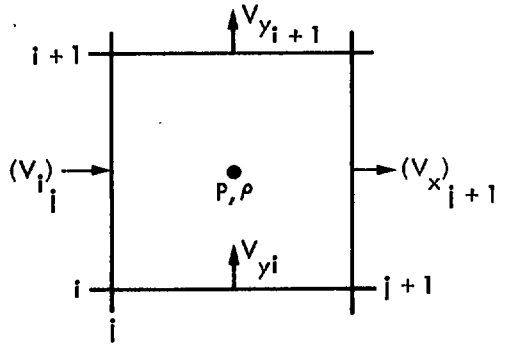

Fig. 4b. Staggered computational mesh

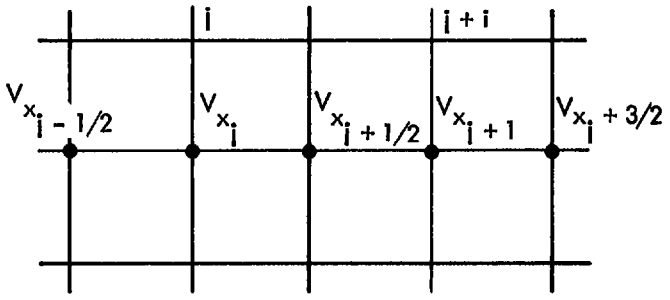

Fig. 5. Velocity fluxes between adjacent cells

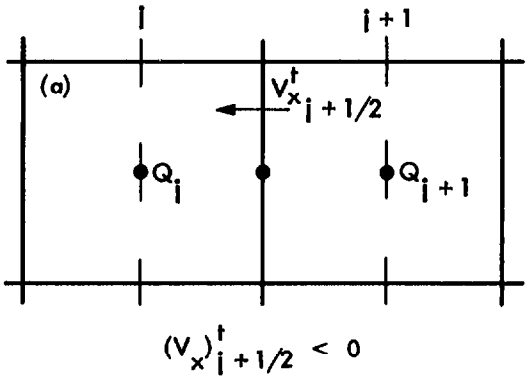

Fig. 6a. Donor cell method, case I

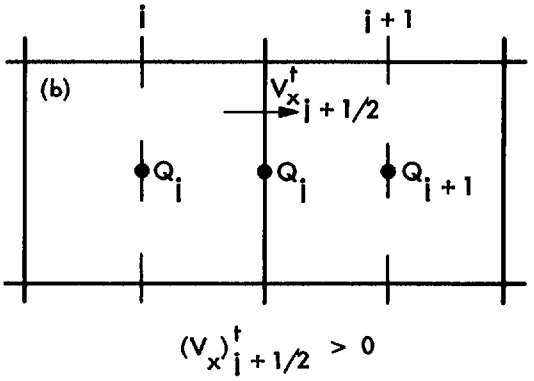

Fig. 6b. Donor cell method, case II

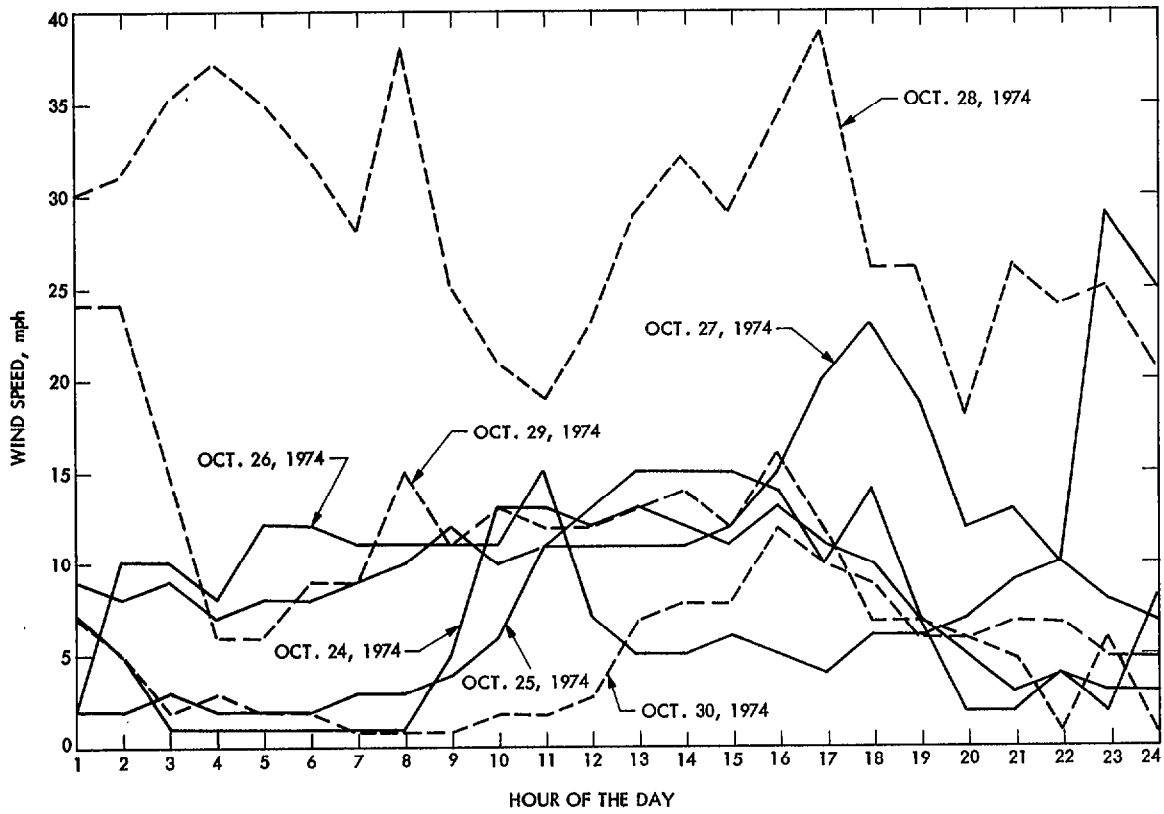

Fig. 7. Wind speed vs time (Goldstone data)

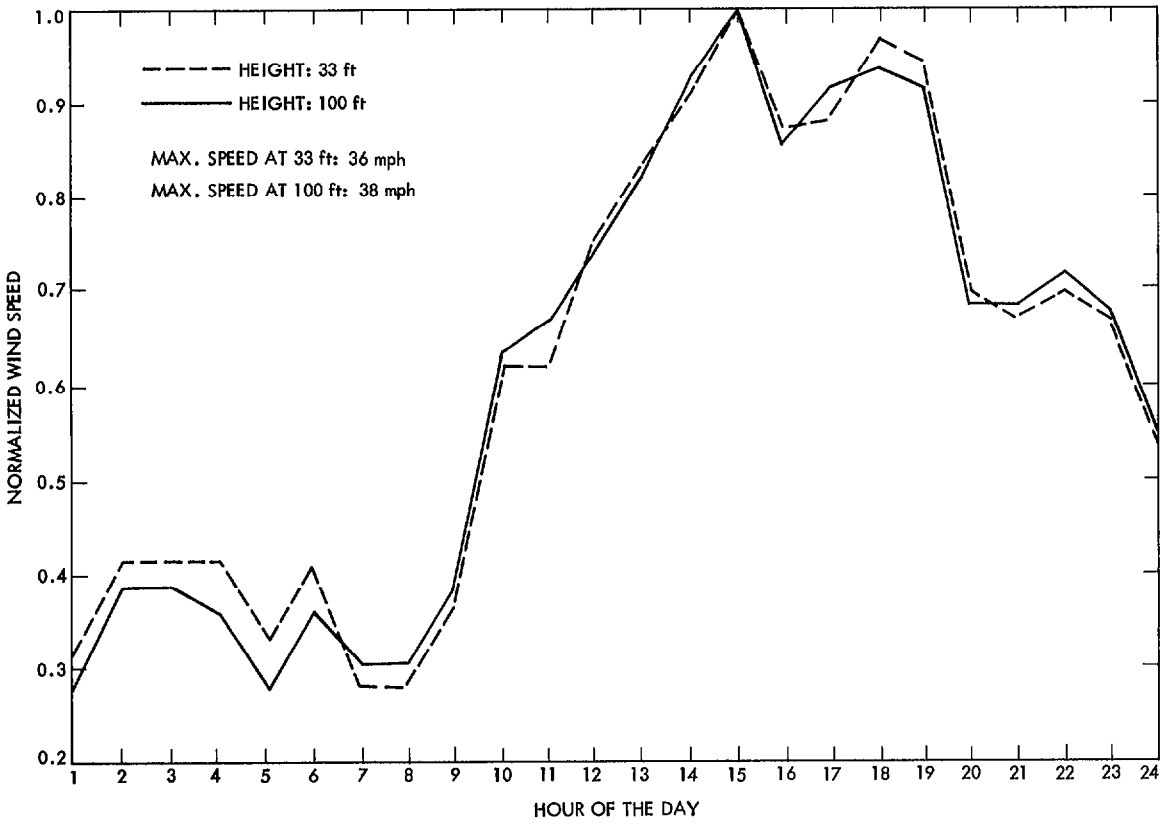

Fig. 8. Comparison of wind speeds at different heights

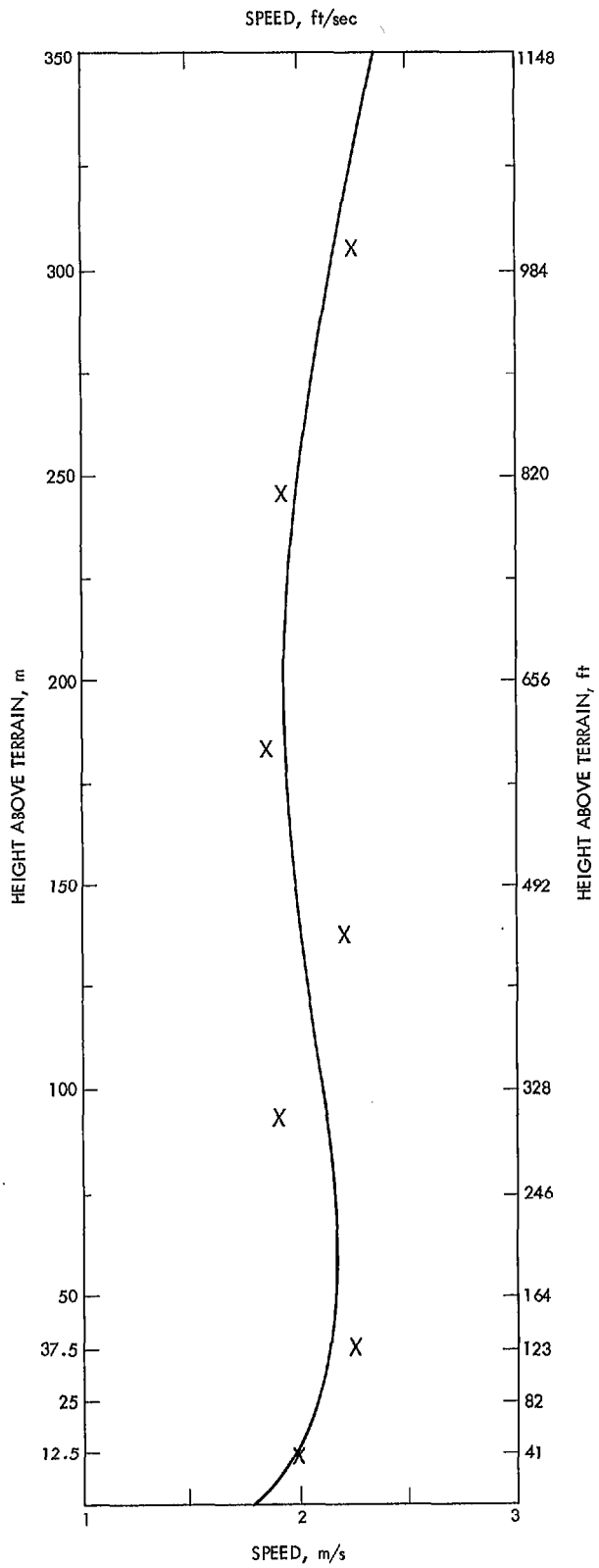

Fig. 9. Wind profile at Savannah River Plant, Ga.

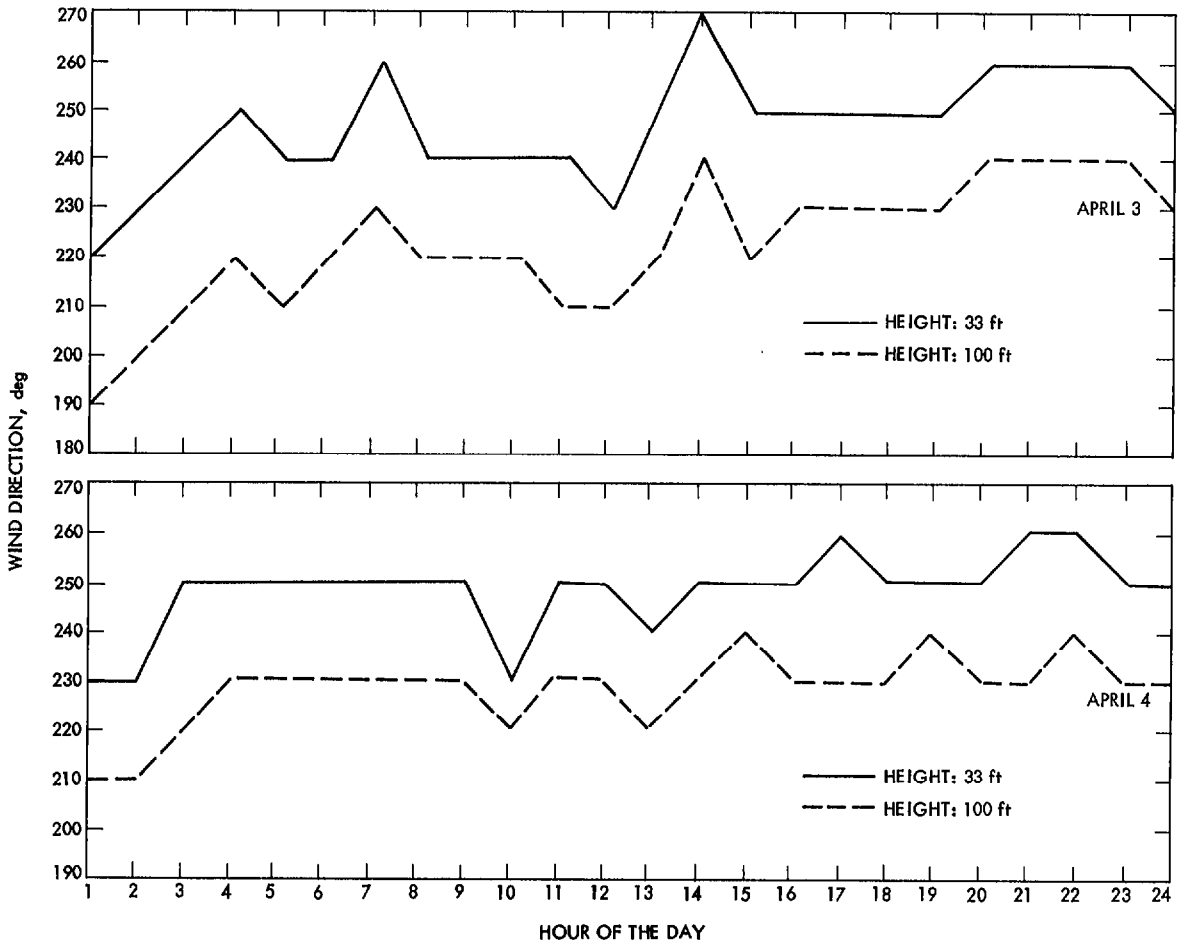

Fig. 10. Comparison of wind directions at different heights

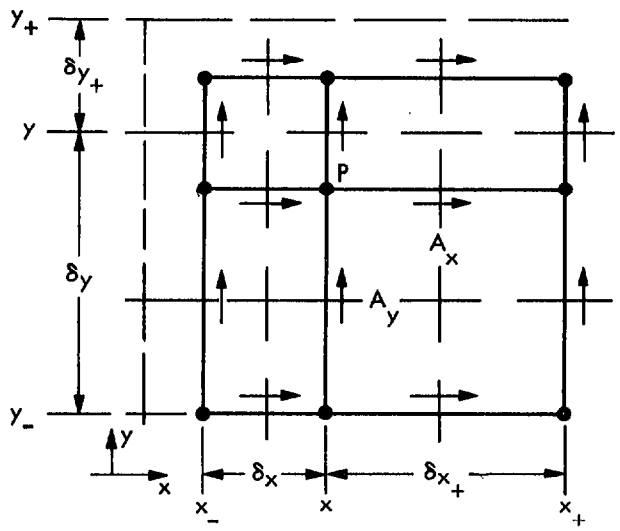

Fig. 11. The staggered grid used in the steady-state model

NASA—JPL—Coml., L.A., Calif.