

Computational analysis of flows

John F. McKibben and Cyrus K. Aidun

ABSTRACT: *The size distribution of black liquor droplets sprayed into a kraft recovery furnace can have a dramatic effect on furnace efficiency and safety. We have developed a computational fluid dynamics tool that allows studying the processes leading to the breakup of a sheet of black liquor into individual droplets. This is accomplished by solving the full Navier-Stokes equations, describing the flow within the black liquor coupled with a potential flow solution in the vapor phase. The results show the accuracy of our technique in predicting the stability of a planar, viscous liquid sheet flowing through a stagnant, inviscid vapor phase.*

KEYWORDS: *Black liquors, computation, drops, equations, fluid dynamics, shear flow.*

The process of spraying black liquor into a recovery furnace is important to the operation and efficiency of the furnace, and particular emphasis must be placed on the distribution of the black liquor droplet sizes. If droplets are too small, they will be entrained in the upward-flowing flue gas, leading to fouling of the heat transfer surfaces. Droplets that are too large may not dry completely prior to reaching the char bed, leading to low bed temperatures and inefficient sulfur reduction.

One typical method used for black liquor spraying involves using a splash plate, shown in Fig. 1. This process involves impinging a jet of black liquor on a flat plate; the liquor is then deflected and spreads

out as a sheet of fluid flowing through the air. Spielbauer (1), in his doctoral work at the Institute of Paper Science and Technology, conducted a study of the mechanisms leading to the breakup of the black liquor sheet into droplets. He found that the initial step in the breakup process results from the growth of waves in the sheet of black liquor. As these waves grow, localized thin spots occur, which spontaneously rupture (due to air bubbles in the black liquor, small nonwetable particles in the black liquor, or some other undetermined cause). These perforations in the sheet then grow due to surface tension forces, intersect with neighboring holes, and form a network of strands. Finally,

the strands become unstable and break up into droplets (1).

We have developed a computational fluid dynamics approach that allows us to study the phenomena of wave growth in the free surface. With the assumptions that the flow is isothermal and incompressible, we begin with the continuity equation:

$$\nabla \cdot \mathbf{v} = 0 \quad (1)$$

arising from the conservation of mass and the Navier-Stokes equation (NSE):

$$\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} = \mathbf{g} - \frac{1}{\rho} \nabla P + \nu \nabla^2 \mathbf{v} \quad (2)$$

In Eq. 2, derived from conservation of momentum, $\mathbf{v}$ is the velocity vector, $\mathbf{g}$ is a body force vector, P is the pressure, ρ is the density, and ν is the kinematic viscosity. These equations are solved in the liquid phase, subject to boundary conditions at the edges of the computational domain, along interior obstacles, and at the interface between the liquid and vapor phases.

To completely describe the hydrodynamic instability leading to sheet breakup, the velocity and pressure fields in the vapor phase must also be computed. With the assumptions that the vapor phase is inviscid and the flow in the vapor phase is irrotational (which it must be if the vapor phase is initially stagnant), the vapor phase may be modeled using potential flow:

$$\nabla^2 \phi_v = 0 \quad (3)$$

where ϕ_v is the vapor phase potential. The pressure and velocities in

McKibben is a doctoral candidate and Aidun is an associate professor at the Institute of Paper Science and Technology, 500 10th St. NW, Atlanta, GA 30338. McKibben is now fluid flow analyst, Procter & Gamble, Cincinnati, OH, 45248.

Recovery Boiler Efficiency

the vapor phase are defined as:

$$P_v = -\rho_v \frac{\partial \phi_v}{\partial t} - \frac{1}{2} |\mathbf{v}_v|^2 \quad (4)$$

and

$$\mathbf{v}_v = \nabla \phi_v \quad (5)$$

where $|\mathbf{v}_v|$ is the magnitude of the vapor phase velocity. Therefore, in the vapor phase, we must solve Laplace's equation, Eq. 3, with Neuman (derivative) boundary conditions in a region with curved boundaries. This is accomplished using standard second-order accurate finite difference formulas for Eq. 3 in the bulk of the vapor phase and adjacent to straight boundaries. At the interface between the two fluids, a more complex treatment is required as we will describe below.

The interfacial boundary conditions are derived from stress and velocity balances. We begin the definition of the stress balances by defining an auxiliary height function, $H(x, y)$, for the location of the interface:

$$H(x, y) \equiv y - \eta(x) = 0 \quad (6)$$

The surface normal vector, $\mathbf{n} = (n_x, n_y)$, is computed from the gradient of $H(x, y)$, which leads to the unit normal vector:

$$n_x = -\eta'(\eta'^2 + 1)^{-1/2}, n_y = (\eta'^2 + 1)^{-1/2} \quad (7)$$

where $\eta' = \partial\eta/\partial x$. The unit tangential vector, $\mathbf{t} = (t_x, t_y)$, is then computed from the orthogonality:

$$t_x = (\eta'^2 + 1)^{-1/2}, t_y = (\eta'^2 + 1)^{-1/2} \quad (8)$$

Finally, the surface curvature, κ , may be computed from the divergence of the unit normal vector as:

$$\kappa = \eta''(\eta'^2 + 1)^{-3/2} \quad (9)$$

With these definitions in mind and the assumption of an inviscid vapor phase, the boundary conditions at the interface arising from normal and tangential stress balances are:

$$P_\ell - \mathbf{n} \cdot \boldsymbol{\tau}_\ell \cdot \mathbf{n} = P_v - \sigma \kappa \quad (10)$$

and

$$\mathbf{t} \cdot \boldsymbol{\tau}_\ell \cdot \mathbf{n} = 0 \quad (11)$$

respectively. The subscripts ℓ and v refer to the liquid and vapor phases, respectively, σ is the surface tension, and:

$$\boldsymbol{\tau} = 2\mu \left(\nabla_{\mathbf{v}} + \nabla_{\mathbf{v}}^T \right) \quad (12)$$

is the viscous stress tensor.

The remaining boundary condition at the interface is due to continuity of velocity. The normal velocity balance is given by:

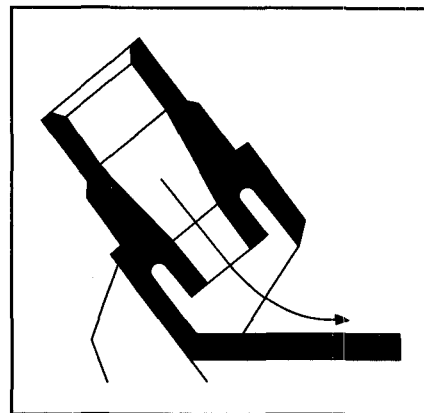
$$\mathbf{v}_\ell \cdot \mathbf{n} = \mathbf{v}_v \cdot \mathbf{n} \quad (13)$$

Since the vapor phase is assumed to be inviscid, continuity of the tangential component of velocity cannot be imposed.

The remaining item needed to completely define the problem is a method for tracking the interface between the liquid and vapor phases. Many methods of tracking are available, and appropriate reviews are easily found (2, 3). In our work, we use the volume-of-fluid (VOF) technique (4) for tracking the location of the liquid phase as it is transported through a fixed Eulerian computational mesh. The VOF method has the advantage of simplicity in treating flows with large deformations and folding free surfaces. In order to increase the range of problems that can be accurately studied with this technique, we have extended the SOLA-VOF method (5) to include the viscous terms in the interfacial boundary condition and, with the assumptions about the vapor phase outlined above, allow flow of the vapor phase. As shown below, the consideration of the vapor phase flow and variations in pressure are critical for stability analysis.

The VOF method is derived from the first generally successful volume-tracking free surface program, the Marker and Cell (MAC) method (6). The MAC method tracks the location of the fluid within a fixed Eulerian mesh by using massless

1. Schematic of a splash-plate black liquor spray nozzle



marker particles. These particles are convected through the computational domain at the end of each time step using the interpolated local fluid velocity. The free surface is then constructed from the cells partially filled with marker particles and having neighboring empty cells. In the MAC method, the normal stress boundary condition at the interface is simplified to:

$$P_\ell = P_v \quad (14)$$

This simplified boundary condition, applied at the cell center rather than the actual interface location, greatly reduced the accuracy of the computational technique.

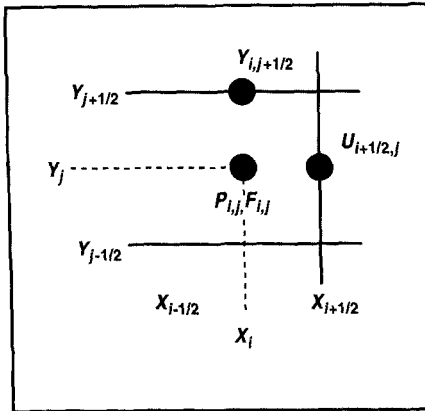
The MAC method has evolved into the VOF technique, which can be considered the limit when the number of marker particles becomes infinite. Thus, the liquid is tracked by a step function, F , representing the fraction of each computational cell occupied by liquid. Transport of F through the computational mesh is governed by the F -convection equation:

$$\frac{\partial F}{\partial t} = \mathbf{v} \cdot \nabla F \quad (15)$$

which ensures that the amount of each phase is conserved.

The interface between the phases is determined on a cell-by-cell basis from local F values. Cells with $F = 1$ are liquid cells, cells with $F = 0$ are

2. Schematic of computational grid geometry



vapor cells, and cells with intermediate F values are free surface cells. Once the free surface cells have been identified, the location and shape of the interface within the free surface cells may be reconstructed from gradients of the F function.

The original VOF implementation (5) included the effects of surface tension, yielding Laplace's formula:

$$P_c = P_v - \sigma \kappa \quad (16)$$

as the interfacial condition. In addition, this technique incorporated an interpolation scheme to apply the boundary condition at the free surface location rather than at the center of the computational cell. Improvements in algorithms for computing surface curvature and methods for treating obstacles within the computational domain were incorporated in a subsequent program (7) followed by extension to three dimensions for cylindrical coordinates (8). All three of these programs neglect the viscous component of the normal stress in the liquid phase at the interface and assume that the pressure in the vapor phase remains constant. These assumptions limit the application of these programs to situations where the viscous component of the interfacial condition is unimportant and where the pressure variations in the vapor phase can safely be neglected.

In the study of free surface flows, such as the die-swell problem (9),

inclusion of the viscous terms in the interfacial boundary condition is vital (10, 11). Therefore, implementations that neglect these components are unable to accurately solve the die-swell problem. In addition, when studying the stability of thin liquid films, the viscous terms at the interface and variations in the vapor phase pressure along the interface are the primary factors inducing instability and wave growth (12).

For these reasons, we have extended the VOF technique to allow inclusion of the viscous terms in the liquid phase at the interface and to allow pressure variation in the vapor phase. We will outline the various solution algorithms used and the implementation of the interfacial condition viscous component. The numerical solution of the vapor phase flow coupled with the liquid phase will be presented with an examination of the combined technique's accuracy.

To test the ability of our method to treat problems involving the stability of two-phase shear flows, we study the stability of a thin liquid sheet flowing through an initially stagnant, inviscid vapor phase. We accurately compute the growth rate of waves in a thin liquid sheet in agreement with predictions from linear stability analysis. This test demonstrates that, despite popular perception, the VOF method can be very accurate and reliable when the complete interfacial conditions are considered.

Numerical technique

We begin describing the numerical technique with a brief outline of the solution algorithm used in the SOLA family of programs. This is followed by a description of the additions we have made to improve the accuracy and extend the capabilities of our program, IPST-VOF3D. Specifically, our additions include more accurate methods for differencing the convective terms in the NSE, inclusion of

the viscous terms in the liquid at the interface, and solution of the potential flow equations in the vapor phase to yield the pressure in the vapor phase.

In the SOLA family of programs, the velocity and pressure fields are solved on a staggered grid (Fig. 2). In this representation, vector quantities are stored on cell faces, and scalar quantities are at the cell centers.

Here, we briefly describe the numerical method, consisting of a two-step projection, used to solve the NSE in the SOLA family of programs. Generally, this can be described by defining an explicit guess:

$$\tilde{\mathbf{v}}^n =$$

$$\mathbf{v}^n + \delta t \left[\mathbf{g} - \frac{1}{\rho} \nabla P^n + \mathbf{v} \nabla^2 \mathbf{v}^n - \mathbf{v}^n \cdot \nabla \mathbf{v}^n \right] \quad (17)$$

for the new velocity field, where the superscript refers to the time step. Except as described below for the convective terms, the specifics of the finite difference representations can be found (5, 7, 8). The velocity field after the time step can then be written as the explicit guess plus a correction due to the pressure change across the time step:

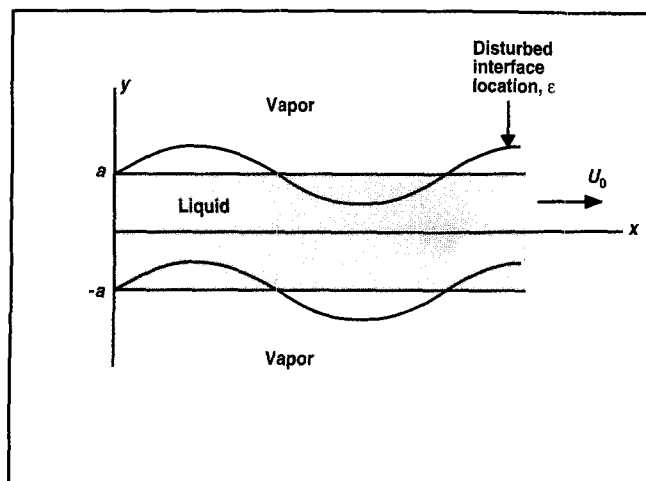
$$\mathbf{v}^{n+1} = \tilde{\mathbf{v}}^n - \frac{\delta t}{\rho} \nabla \left(\delta P^{n+1} \right) \quad (18)$$

Since mass must be conserved at all times, we may substitute Eq. 18 into Eq. 1, yielding:

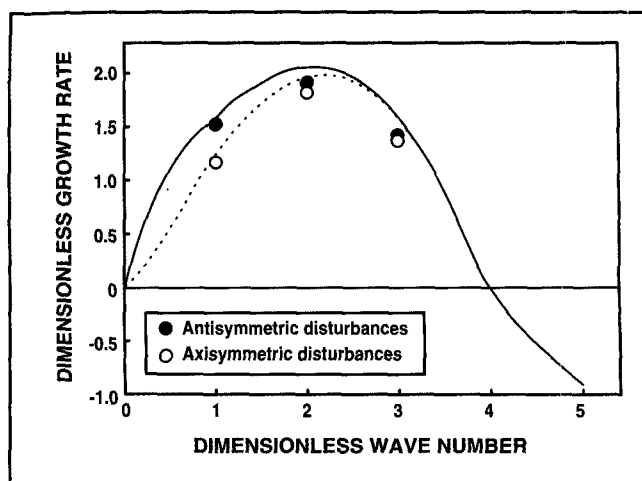
$$\frac{\delta t}{\rho} \nabla \cdot \left[\nabla \left(\delta P^{n+1} \right) \right] = \nabla \cdot \tilde{\mathbf{v}}^n \quad (19)$$

where V is the volume of the computational cell needed to ensure a symmetric system of equations (7). The Poisson equation for pressure in Eq. 19 yields a sparse, symmetric, linear system that can be solved using a variety of numerical methods such as successive over-relaxation (SOR) or the conjugate residual (CR) methods (8). With the new pressure field

3. Schematic of sheet instability problem



4. Nondimensional growth rate for $We_c = 40$, $Z = 0.1$, and $\bar{\rho} = 0.1$, obtained from numerical solution of Li and Tankin's (12) dispersion relations. Open and closed circles represent results of our computational analysis.



available, the corrected velocity field is then computed from Eq. 18.

The F -convection equation, Eq. 15, is solved using donor-acceptor differencing (8) to assist in maintaining a sharp interface between the liquid and the vapor phases. Once the new fluid configuration has been obtained, it is possible to reconstruct the localized interface configuration needed to compute the surface tension force (8). Again, the details of this process are presented elsewhere (5, 7, 8).

As we have shown previously (9) and as the Reynolds number increases, the accuracy of the finite difference representation of the convective terms in the NSE limits the accuracy of the entire solution. Therefore, in addition to the standard differencing for the convective terms present in the SOLA programs, we applied and evaluated three third-order accurate differencing options: quadratic upstream interpolation for convective kinematics (QUICK) (13, 14), third-order accurate upwind differencing (THIRD) (3, 15), and the method of Kawamura and Kuwahara (KANDK) (3, 16). The details of these differencing techniques are presented elsewhere (3, 9).

The previous VOF techniques

used Laplace's formula, Eq. 16, as a simplified way to calculate the normal stress balance. To eliminate the assumption inherent in Eq. 16 that the viscous terms in the interfacial boundary condition are negligible, we have included an option for computing the viscous forces. The local unit vector normal to the interface is computed in the manner described above. Once the coordinate axis most nearly normal to the interface has been determined, a local height function analogous to Eq. 6 is computed, and the unit surface normal is obtained from Eq. 7.

Next, the components of the viscous stress tensor, Eq. 12, are computed using the provisional velocities, $\tilde{\mathbf{v}}^n$, where only velocities within the liquid phase are included in the finite difference formulas. Finally, the viscous component of the interfacial boundary condition is computed from $\mathbf{n} \cdot \boldsymbol{\tau}_i \cdot \mathbf{n}$.

Potential flow in the vapor phase

As previously stated, for stability problems, such as flow of a thin liquid sheet, allowing the pressure in the vapor phase to vary is vital. We have implemented a method for solv-

ing the potential flow equation in the vapor phase that is coupled to the full NSE in the liquid phase through the interfacial conditions. This allows computation of the pressure in the vapor phase as a function of time and position.

The major challenge in solving the vapor phase potential equation, Eq. 3, is imposing the Neuman boundary conditions along the curved boundary made up of the interface between the liquid and vapor phases. We have implemented a modified form of Bramble and Hubbard's second-order accurate method (17) for solving Poisson's equation in a region with curved boundaries having mixed boundary conditions. Bramble and Hubbard define a second-order accurate operator as:

$$\delta_n \phi_0 = \phi_0 \sum_{i=1}^3 a_i - \sum_{i=1}^3 a_i \phi_i \quad (20)$$

for the normal derivative using three points within the region of interest (in this case, the vapor phase). The coefficients a_i are determined from solving the system of equations:

$$\text{See Equations} \quad (21)$$

where $\bar{y}$ and $\bar{x}$ are the distances from the surface point to neighboring points within the flow in the normal

and tangential directions, respectively. In addition, Bramble and Hubbard present criteria that, when satisfied, ensure that the operator yields a diagonally dominant system of equations.

The boundary condition for the vapor phase potential at the interface is conservation of the normal velocity, Eq. 13. Thus, the boundary operator is equal to the normal velocity in the liquid phase plus a small correction arising from the derivation:

$$\delta_n \phi_0 = v_{\bar{y}} + \frac{\partial v_{\bar{y}}}{\partial \bar{x}} \sum_{i=1}^3 a_i \bar{x}_i \bar{y}_i \quad (22)$$

where $v_{\bar{y}}$ is the normal velocity at the interface and $\partial v_{\bar{y}} / \partial \bar{x}$ is the tangential derivative of the normal velocity at the interface (17). Again, since we have assumed that the vapor phase is inviscid, no restrictions are placed on the tangential velocity at the interface.

By incorporating the liquid phase viscous terms and variations in the vapor phase pressure in the interfacial boundary condition, the solution procedure becomes:

- Compute the surface curvature from the local liquid configuration.
- Compute the surface normal velocity from the change in surface position.
- Solve Eq. 3 for the vapor phase potential.
- Compute the vapor phase pressure from Eq. 4.
- Compute the explicit guess for the liquid phase velocity field from Eq. 17.
- Compute the interfacial liquid phase stress from Eq. 12.
- Compute the pressure on the liquid side of the interface from Eq. 16.

- Solve the Poisson pressure equation, Eq. 19, yielding the liquid phase pressure field.
- Update the liquid phase velocity field using Eq. 18.
- Solve Eq. 15 to yield the new fluid configuration.

These steps may be repeated until the desired time is reached. The second and third steps have been added to allow for variations in the vapor phase pressure, and the fifth step is required for including the vapor phase viscous terms in the interfacial conditions.

Results and discussion

To test the implementation of the coupled inviscid vapor phase with the full NSE in the liquid phase, we study the stability of a thin liquid sheet flowing through an inviscid vapor phase, as presented in Fig. 3. The variation in the surface position, ϵ , shown in Fig. 3 is assumed to be of the form:

$$\epsilon = \epsilon_0 e^{\omega t + i k x} \quad (23)$$

where ϵ_0 is the initial amplitude, $\omega = \omega_r + i \omega_i$ is the complex growth rate, $i = -1^{1/2}$, and k is the wave number of the disturbance. A study using linear stability analysis (12) yields dispersion relations between the wave number and the growth rate for antisymmetric disturbances:

$$\text{See Equations} \quad (24)$$

and axisymmetric disturbances:

$$\text{See Equations} \quad (25)$$

where

$$\tilde{\omega} = \tilde{\omega}_r + i \text{We}_l^{1/2} \tilde{\omega}_i \quad (26)$$

$$\tilde{\omega}_1 = \tilde{\omega} + i \text{We}_l^{1/2} m \quad (27)$$

$$\tilde{\omega}_r = \omega_r \left(\rho_l a^3 / \sigma \right)^{1/2} \quad (28)$$

$$\tilde{\omega}_i = \omega_i \left(a / U_0 \right) \quad (29)$$

and a is the initial sheet half-thickness, $m = ka$ is the dimensionless wave number, and U_0 is the initial sheet velocity. The remaining parameters are defined as the Weber number:

$$\text{We}_l = \rho_l U_0^2 a / \sigma \quad (30)$$

the Ohnesorge number:

$$Z = \mu_l \left(\rho_l a \sigma \right)^{-1/2} \quad (31)$$

and the density ratio:

$$\bar{\rho} = \rho_v / \rho_l \quad (32)$$

It is possible to numerically solve the dispersion relations for given values of We_l , Z , and $\bar{\rho}$ to yield the complex growth rate, $\tilde{\omega}$, as a function of the dimensionless wave number, m . The real part of $\tilde{\omega}$ is the dimensionless growth rate of a disturbance with wave number m . Results for $\text{We}_l = 40$, $Z = 0.1$, and $\bar{\rho} = 0.1$ are shown in Fig. 4 for both antisymmetric and axisymmetric disturbances, represented by solid and dashed lines, respectively. The data points plotted in Fig. 4 represent our computed results, with the filled circles resulting from antisymmetric disturbances and the open circles from axisymmetric disturbances.

Conclusions

The instability leading to the breakup of a black liquor sheet is an important step in the droplet formation process. We have developed a computational technique, which is based on the solution of the full Navier-Stokes equations in the liquid phase coupled with a potential flow solution in the vapor phase. This technique can be used to study the physical processes leading to this instability. **□**

Recovery Boiler Efficiency

Literature cited

1. Spielbauer, T. M., "The stability and disintegration of radially thinning liquid sheets," Doctoral dissertation, Institute of Paper Science and Technology, Atlanta, 1992.
2. Floyryan, J. M. and Rasmussen, H., *Applied Mechanics Reviews* 42(12): 323(1989).
3. McKibben, J. F., "A computational fluid dynamics model for transient three-dimensional free surface flows," Doctoral dissertation in progress, Institute of Paper Science and Technology, Atlanta, 1993.
4. Hirt, C. W. and Nichols, B. D., *Journal of Computational Physics*, 39: 201(1981).
5. Nichols, B. D., Hirt, C. W., and Hotchkiss, R. S., Los Alamos Report LA-8355, Los Alamos National Laboratory, Los Alamos, NM, 1980.
6. Harlow, F. H. and Welch, J. E., *Physics of Fluids* 8(12): 2182(1965).
7. Torrey, M. D., Cloutman, L. D., Mjolsness, R. C., et al., Los Alamos Report LA-10612-MS, Los Alamos National Laboratory, Los Alamos, NM, 1985.
8. Torrey, M. D., Mjolsness, R. C., and Stein, L. R., Los Alamos Report LA-11009-MS, Los Alamos National Laboratory, Los Alamos, NM, 1987.
9. McKibben, J. F. and Aidun, C. K., *TAPPI 1991 Engineering Conference Proceedings*, TAPPI PRESS, Atlanta, p. 711.
10. Hill, G. A., "Numerical simulation of free surface flows," Doctoral dissertation, University of Saskatchewan, Saskatoon, SK, Canada, 1979.
11. Hill, G. A., Shook, C. A., and Esmail, M. N., *Canadian Journal of Chemical Engineering* 59: 100(1981).
12. Li, X. and Tankin, R. S., *Journal of Fluid Mechanics* 226: 425(1991).
13. Leonard, B. P., *Computer Methods in Applied Mechanics and Engineering* 19: 59(1979).
14. Freitas, C. J., Street, R. L., Findikakis, A. N., et al., *International Journal for Numerical Methods in Fluids* 5(6): 561(1985).
15. Agarwal, R.K., American Institute of Aeronautics and Astronautics Paper AIAA-81-0112, AIAA, New York, 1981.
16. Kawamura, T. and Kuwahara, K., American Institute of Aeronautics and Astronautics Paper AIAA-84-0340, AIAA, New York, 1984.

Equations

$$\begin{bmatrix} \bar{y}_1 & \bar{y}_2 & \bar{y}_3 \\ \bar{x}_1 & \bar{x}_2 & \bar{x}_3 \\ \bar{x}_1^2 - \bar{y}_1^2 & \bar{x}_2^2 - \bar{y}_2^2 & \bar{x}_3^2 - \bar{y}_3^2 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \end{bmatrix} = \begin{bmatrix} 1 \\ 0 \\ 0 \end{bmatrix} \quad (21)$$

$$0 = (\bar{\omega}_1 + 4m^2 Z) \omega_1 \tanh(m) + \bar{\rho} \bar{\omega}_1^2 + m^3 + 4m^3 Z \left[m \tanh(m) + \sqrt{m^2 + \bar{\omega}_1 / Z} \tanh \sqrt{m^2 + \bar{\omega}_1 / Z} \right] \quad (24)$$

$$0 = (\bar{\omega}_1 + 4m^2 Z) \bar{\omega}_1 \coth(m) + \bar{\rho} \bar{\omega}_1^2 + m^3 + 4m^3 Z \left[m \coth(m) + \sqrt{m^2 + \bar{\omega}_1 / Z} \coth \sqrt{m^2 + \bar{\omega}_1 / Z} \right] \quad (25)$$

17. Bramble, J. H. and Hubbard, B. E., *Journal of the Association of Computing Machinery* 12(1): 114(1965).

This study is supported by industrial contributions and the National Science Foundation under Grant CTS-9258667. The computations were conducted, in part, using the Cornell National Supercomputer Center, a resource of the Center for Theory and Simulation in Science and Engineering at Cornell University, which is funded in part by the National Science Founda-

tion, New York State, and the IBM Corp. Portions of this work were used by the author as partial fulfillment of the requirements for a Ph.D. degree at the Institute of Paper Science and Technology.

Received for review June 30, 1993.

Accepted Sept. 30, 1993.

Presented at the TAPPI 1993 Engineering Conference.