Visualization of the recovery-boiler flow fields predicted by computational fluid dynamics

Paul J. Chapman, Stephen G. Janik, and Andrew K. Jones

Design engineers can animate simulated flow fields in the recovery furnace using the latest visualization software.

Flow patterns in the kraft recovery furnace can be simulated using models based on computational fluid dynamics (CFD). The use of CFD is becoming increasingly common as computer workstations become more powerful and CFD software is improved.

Models based on Fluent code (*1*) include the major physical and chemical processes occurring inside the recovery furnace (*2*). However, the task of accurately representing black-liquor combustion with the various char-bed and gas-phase reactions is very complex. Models with greater detail are needed for recovery-furnace simulations. To this end, the Department of Energy is sponsoring efforts to develop and validate computational-based black-liquor combustion models (*3*).

The main reason for using CFD to simulate the flow fields in a kraft recovery furnace is that it saves time and money. Mathematical models allow engineers to examine the effects of system changes without going through the expensive and painstaking process of directly measuring flows within the furnace. Computer predictions of gas velocity and mixing patterns are reliable and have been validated with test data from physical models and from the field.

The conventional way of interpreting CFD results is by plotting vectors and contours. This article examines the use of scientific visualization (SV) software to analyze CFD predictions. SV software such as NASA's PLOT3D and FAST (*4*, *5*) produces detailed images of CFD results, providing system designers with a comprehensive view of how flow fields are affected by changes in design. SV tools are equally useful for detailed isothermal flow-field predictions as well as more-complex simulations that include combustion models with heat transfer.

Chapman is principal engineer and Janik is a programmer/analyst in the Kreisinger Development Laboratory of ABB Combustion Engineering Systems, 1000 Prospect Hill Rd., Windsor, CT 06095-1521. Jones is principal engineer, research and development, for ABB Canada, 99 Bank St., Ottawa, ON, Canada K1P 6C5

In this article, we present simulated results for flow fields in the lower furnace. Because the flows in the lower furnace are dominated by the air system, we chose to simulate flow fields under isothermal conditions. The predicted flow fields were used to supplement results obtained from physical modeling.

When a physical model is used for testing, each air-system configuration is typically evaluated based on air and gas velocities and the mixing distribution as measured at a limited number of test planes. Such measurements are commonly used to quantitatively assess air-system configurations for modeling studies or to validate CFD models. However, the task of optimizing a recovery-furnace air system requires knowledge of the flow fields throughout the entire lower furnace region. Only then can the designer perceive the global effects of air-system changes. The flow field in the zone where liquor droplets swell and devolatilization occurs is critical to assessment of carry-over and heat-release rates in the lower furnace. This underscores the need to develop and validate comprehensive CFD combustion models.

Current developments

Using CFD to predict turbulent flow fields remains a challenging task. CFD predictions are obtained using iterative solver techniques that can require enormous amounts of time for convergence. The traditional method of performing CFD involved grid preprocessing and required manual input or, in some cases, mesh generators. Short-term success rates were marginal and required long-time commitments to obtain converged CFD solutions. These tasks were delegated to mainframe computers, where they were typically carried out as batch jobs. The CFD results were then downloaded to minicomputers for post-processing routines, which involved plotting the results as vectors or contours.

The availability of sophisticated tools for CFD analysis has lagged behind other disciplines such as structural analysis and conceptual product design. However, with recent improvements in both the CFD codes and increasingly powerful minicomputers, the use of CFD is growing rapidly. With the introduction of CFD codes that can be run on workstations, traditional CFD flow-field analysis using conventional vector and contour plotting is often inadequate. Better data-presentation techniques are needed for both CFD and physical models. Visualization software is now available to fill this void. These packages can interface with commercial CFD codes to provide powerful reviewing techniques.

New technology

CFD for industrial products is migrating from the mainframe environment to the graphics workstation. With the rapid increase in workstation capabilities, larger CFD simulations can be performed at the local level. The engineer can now run larger problems locally, performing both pre- and post-processing as well as computation tasks on the same machine. Detailed CFD simulations of isothermal and reacting flow with heat transfer are becoming economically viable. With the expanding use of CFD, there is a clear need for better software to analyze CFD results.

CFD vendors are competing to capitalize on the expanding graphics hardware capabilities to generate the most appealing visualization interface. This will probably be achieved by teamwork between the code vendors, the developers of visualization software, and hardware vendors. As Windows®-based 3-D graphics standards evolve, software developers will no longer have to modify their products to support multiple hardware platforms. This will streamline the process of developing new software, benefitting everyone. The target device for most visualization packages is the 3-D graphics workstation (*6*). These software packages are relatively flexible and accept a range of engineering data. Until recently, this sophistication was unavailable.

Simulations of recovery-furnace combustion and heat transfer are also becoming increasingly sophisticated. Analysis of steady-state and dynamic simulations can be enhanced enormously through the use of available visualization tools. Before selecting a particular package, however, the user must establish the visualization requirements for the application and evaluate the available packages based on these criteria.

Determining visualization requirements

Visualization is becoming an increasingly vague term, so it is up to the user to define what it means. Before selecting a visualization package for analysis of CFD predictions, the user should be familiar with the various visualization options. Many packages are tailored to provide specific capabilities. The new graphics options extend beyond the traditional presentation techniques, such as static vector and contour plotting, into the realm of 3-D rendering, animation, removal of hidden surfaces, enhancement, real-time translation, and zoom capability. These powerful display techniques put the designer on top of the tedious results-evaluation task.

Understanding the visualization requirements can help simplify the process of selecting an SV package. Some of the main options influencing the choice are as follows:

- How will the data be viewed or enhanced?
- Are global results preferred to local enhancement?
- Is a static image acceptable, or is animation essential?
- Are multiple views or simulations to be evaluated simultaneously?
- Are results to be transferred to other media?

After answering these questions, the engineer should review the capabilities of each SV package to determine the best choice. Several packages have been reviewed in the literature (*7*).

Interface between CFD and SV

Transferring the results from the CFD code to the visualization package is usually a one-time task of establishing a link between the software and the CFD data files using the proper format. Visualization software is often marketed with the implication that establishing an interface is a simple task. Recent experience suggests that this is not always true. As the size of the data files increases, an efficient link between the grid geometry and data files is essential for tasks such as animation. Responding to customer demands, several CFD vendors are in the process of developing standards for a universal interface.

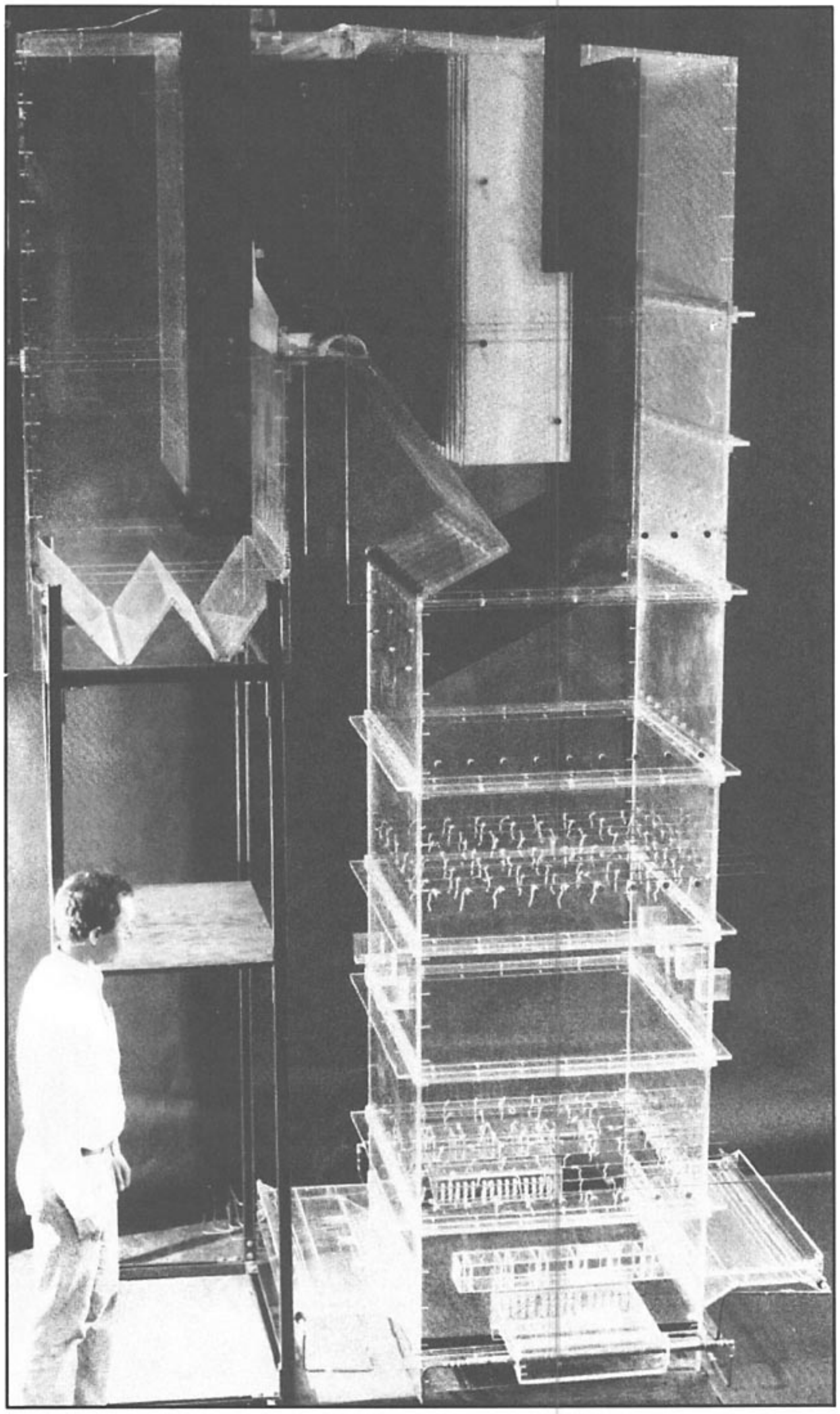
CFD simulation of recovery furnace

To illustrate the utility of SV, we simulated the flow fields in a recovery furnace under isothermal conditions using Creare.x's Fluent code (*1*). The simulated CFD case is a 1/8-scale recovery-boiler model constructed and tested at our laboratory in Windsor, CT. The model shown in **Fig. 1** has been used for a variety of secondary-air design tests using 3-D velocity mapping and methane gas mixing tests, as detailed in the literature (*8, 9*).

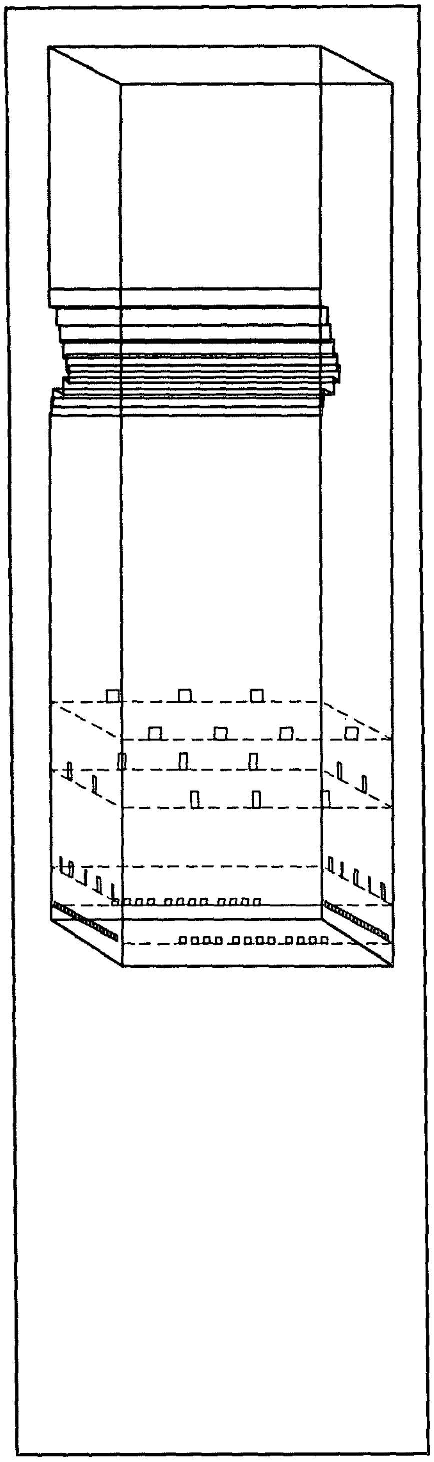
The CFD outline in **Fig. 2** represents the furnace from the floor through the vertical outlet plane and shows the various injection locations. The furnace was represented using 202,000 nodes, 88 in the vertical direction and a 48 × 48-node mesh at each horizontal slice. Although side symmetry conditions could be used for some arrangements, the choice to model the entire furnace was preferred because it allowed us to observe the effects of interlaced as well as opposed secondary air. The nodal distribution in the vertical direction concentrated the nodes in the lower furnace region, where flow-field gradients and mixing levels were highest.

Initial CFD velocity results compared reasonably well with the five-hole velocity measurements at the lower test plane, indicated by the yarn tufts in the physical model. For the flow model, delivered air flows included primary, secondary, and tertiary air. However, the CFD predictions redistributed the additional input sources to better account for the difference between air and gas flows in the boiler. These contributions were represented as a uniform superficial vertical gas velocity across the bed surface, simulating volatile gases. The remaining portion of the liquor flow and infiltration were injected at the gunport level.

1. Isothermal scale model of recovery furnace



2. Computational model with injection ports for primary, secondary, and tertiary air



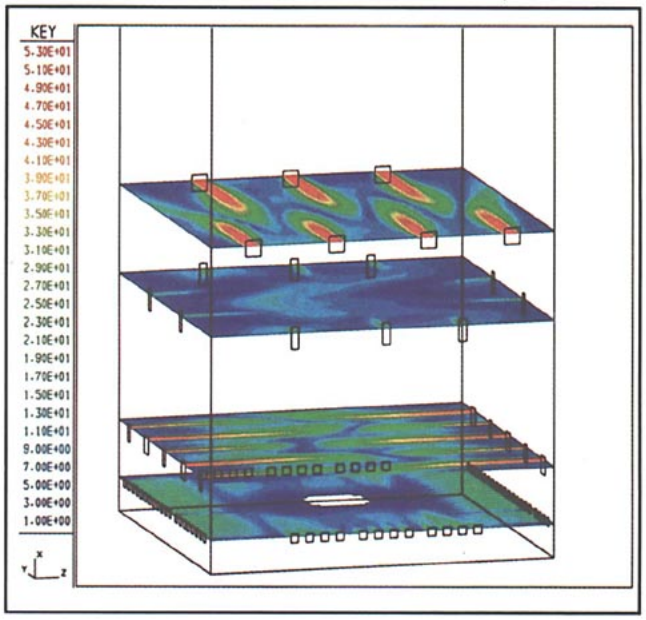
Visualization

The influence of secondary-air placement on the predicted gas flow fields was studied using visual comparisons. In the case presented here, we present the predicted velocity distributions in the lower furnace for a two-wall secondary-air system with interlaced tertiary air. Injection at each of the air levels is shown in **Fig. 3**, with the color spectrum from red to blue corresponding to a high-to-low velocity distribution.

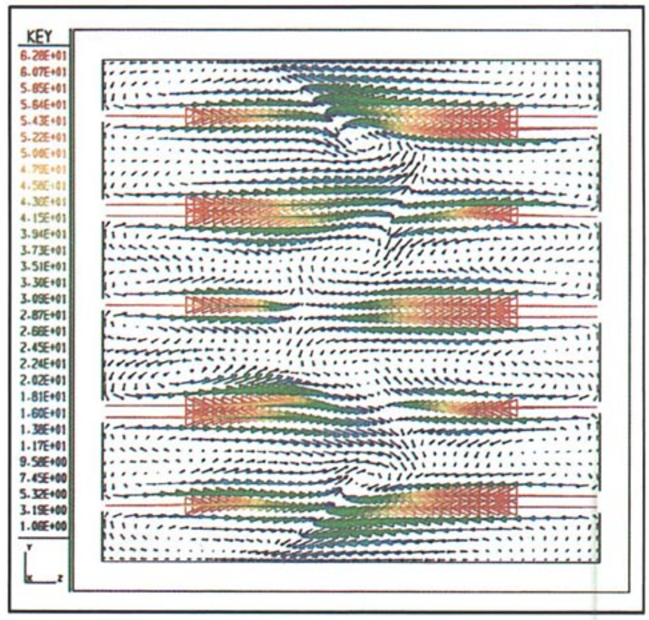
Velocity vectors at the plane of secondary-air injection are shown in **Fig. 4**. The penetration of the secondary-air jets in this configuration of alternating large and small ports is clearly illustrated. This configuration was judged to provide improved vertical velocity distribution and good mixing characteristics.

The conditions observed in different slices can vary considerably, as seen in **Figs. 5 and 6**, which show the

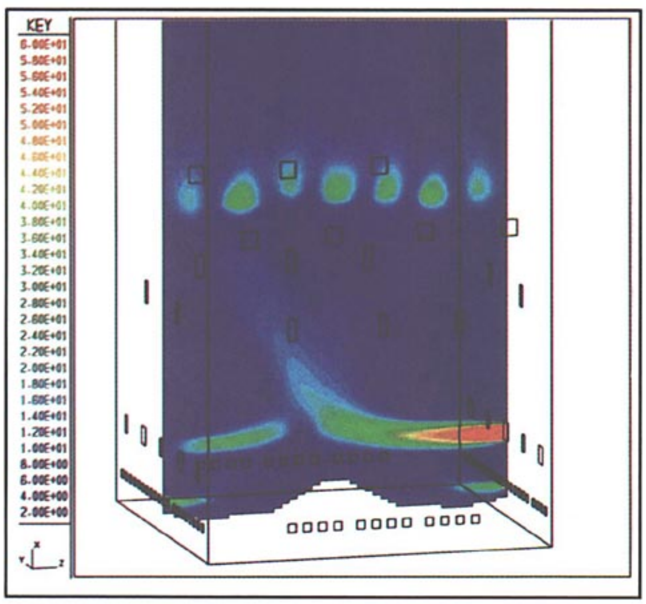
3. Injection velocities in lower furnace. The Key column represents velocity in units of meters per second, with red indicating the highest velocity and blue the lowest. (E + 01 = 10^1; E - 01 = 10^1)



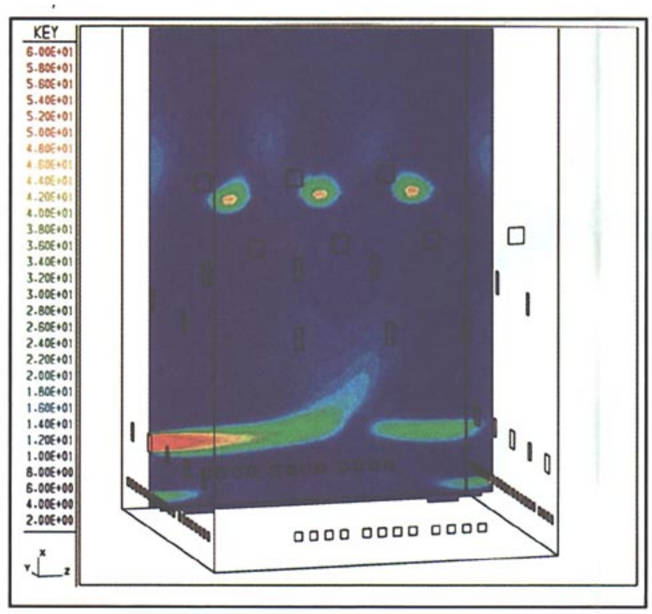
4. Vectors for injection velocity of secondary air. (E + 01 = 10^1; E - 01 = 10^1)



5. Injection velocity at centerline slice. (E + 01 = 10^1; E - 01 = 10^1)



6. Injection velocity at slice closer to rear wall. (E + 01 = 10^1; E - 01 = 10^1)



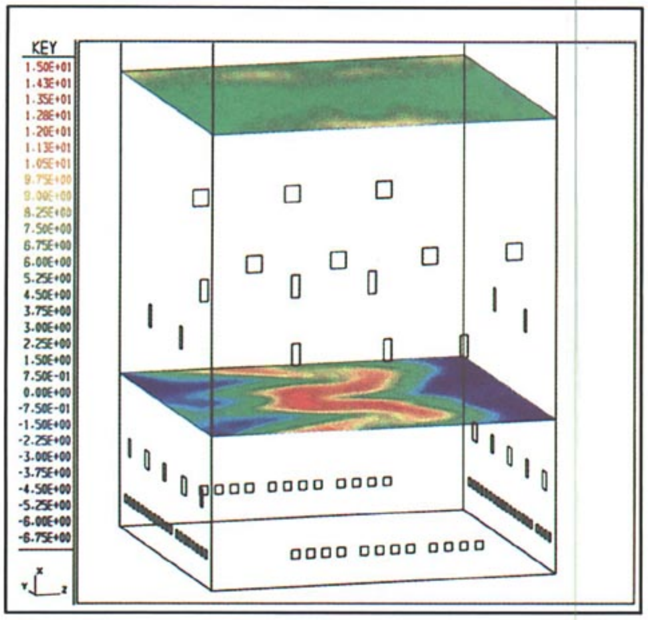
jet trajectories at two consecutive ports. One critical aspect in optimizing recovery-boiler performance is the thorough mixing of oxygen throughout the cross-sectional area so that all combustible gases are burned. The particular arrangement and operating characteristics of the secondary-air ports play a dominant role.

In addition to the mixing requirements, the vertical velocity distribution is important for limiting particulate carry-over. Vertical velocity distribution is largely influenced by air-system variables as well as by furnace geometry. The predicted vertical velocity distributions in **Fig. 7** correspond to the test planes identified by the arrays of yarn streamers in the physical model in Fig. 1. At the lower test plane, central upward velocity with recirculation zones above the secondary-air sidewall nozzles is observed. From these studies, it has been found that the maximum central-core velocities and recirculation zones can be reduced but not eliminated. At the test plane above the tertiary air, the velocity distribution is relatively uniform.

These computed results show the overall velocity distribution, providing a detailed description of the steady-state turbulent flow field. Using CFD, design engineers can efficiently evaluate several different configurations, thus streamlining the design process for air systems.

Clearly, the three-dimensional character of the flow field requires viewing the data at more than a few selected

7. Upward velocity within two horizontal slices. (E + 01 = 10^{1}; E - 01 = 10^{1})



planes. By importing the Fluent (*1*) predictions into the FAST code (*5*), the design engineer can create sweeping vector or raster displays of velocity or species concentration over the various coordinate directions. Although it is difficult to illustrate here, this capability is extremely useful in understanding and quantifying the various design configurations being simulated.

Animation is best displayed on a high-resolution graphics monitor. The animation sequence can be transferred to video or film using additional hardware. Use of scan converters to commercial television format is useful for industrial applications, but some loss in clarity is experienced using standard TV resolution. Frame-by-frame digitizing techniques are often used to improve the transfer quality. At this point in time, film is the preferred transfer medium (*10*).

Conclusion

Engineers are increasingly relying on CFD (computational fluid dynamics) models to design and optimize recovery-boiler air systems. The computed flow fields are usually plotted using vectors or contours. However, the graphics capabilities embodied in most CFD codes can be exploited more fully using one of the commercially available visualization software packages. The new graphics options extend the traditional vector and contour plots into the realm of 3-D portrayals, with the ability to animate images, remove surfaces, and zoom in for close-up views.

As CFD models become more sophisticated, simulations of recovery-boiler flow fields are becoming increasingly complex. Engineers are finding that visualization software is a powerful tool for tracking the intricate changes that occur in flow fields when design parameters are modified.□

Literature cited

1. Fluent user's manual (Version 3.03), Creare.x, Hanover, NH.
2. Jones, A., A model of the kraft recovery furnace, Ph.D. thesis, Institute of Paper Chemistry, Appleton, WI, 1988.
3. Black liquor chemical recovery research, U.S. Dept. of Energy, in Proceedings of 1991 Program Review Meeting, U.S. DOE, Washington, DC.
4. PLOT3D software for visualization of grids and solutions to CFD, NASA Ames Research Center, distributed by COSMIC, University of Georgia, Athens, Program ARC-12782, March 1990.
5. FAST (Flow Analysis Software Toolkit) user's manual, Watson, V. *et al.*, Sterling Federal Systems, NASA Ames Research Center, Moffett Field, CA, 1990.
6. Carrabine, L., Computer-Aided Eng. 10(1): 19(1991).
7. Carrabine, L., Computer-Aided Eng. 10(2): 6(1991).
8. Chapman, P., and Jones, A., 1990 Engineering Conference Proceedings, TAPPI PRESS, Atlanta, p. 193.
9. Jones, A., Chapman, P., and Mahaney, J., 1991 Engineering Conference Proceedings, CPPA, Montreal, p. A123.
10. Snuggerland, K., Silicon Graphics Magazine of Visual Processing 14:(1991).

Received for review July 31, 1991.

Accepted Oct. 6, 1991.

Presented at the 1991 TAPPI Engineering Conference

Keywords: Air flows, boilers, fluid dynamics, fluid flow, graphic methods, recovery furnaces, simulation.