MODELLING SOOTBLOWER JET EFFECTIVENESS WITH ANSYS FLUENT

SHAHED DOROUDI, AMEYA POPHALI, MARKUS BUSSMANN*, DANNY TANDRA, HONGHI TRAN

ABSTRACT

Sootblower jet effectiveness is a strong function of the force exerted by a jet on a fireside deposit, which in turn depends strongly on the local tube geometry within a recovery boiler. Past research at the University of Toronto, both experiments and simulations, has examined the dynamics of sootblower jet interaction with tube geometries in the superheater, but the CFD (computational fluid dynamics) studies used a research code that was difficult to apply to complex geometries. Recently, turbulence model corrections developed during that time have been incorporated into ANSYS Fluent, making it possible to develop more complex models of jet/tube interaction in the generating bank and economizer sections. This paper presents an overview of our latest work on developing those models, in particular our attempts to identify an appropriate inlet boundary condition at the sootblower jet nozzle. This research has yielded both Mach number and pressure distributions within an off-design jet that agree well with experimental data.

INTRODUCTION

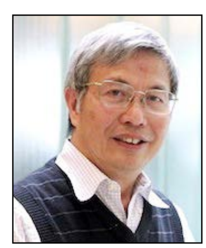
Sootblowers are used to remove fireside deposits from heat-transfer tube surfaces in Kraft recovery boilers by using high-pressure boiler steam to generate a pair of supersonic jets that are directed at deposits. Sootblowing effectiveness is directly related to the jet force exerted on deposits during blowing. Sootblowers can consume 10% or more of the steam generated by a boiler and therefore represent a significant cost. This motivates research into optimizing sootblowing by maximizing deposit removal and minimizing steam usage.

Over many years, research at the University of Toronto has examined many aspects of sootblower operation, using both experiments and CFD modelling, as summarized in a recent overview [1]. The modelling research was originally conducted using the CFDLib code developed at the Los Alamos National Laboratory. Tandra [2,3] investigated fully expanded jets, including a study of the effectiveness of low-pressure sootblowing; Emami [4,5] extended the work to off-design (over- and under-expanded) jets impinging on geometries characteristic of superheater platens in recovery boilers. Recently, the authors began to use the ANSYS Fluent commercial CFD software to model jet/tube interactions in more complex geometries like those of the generating bank [6] and comparing the results to schlieren images and pitot-tube measurements of air jet flow into one-quarter scale models of recovery boiler superheater, generating bank, and economizer geometries [7].

This paper reports on recent work using ANSYS Fluent to model sootblower jet flow into an economizer geometry and on the challenge of obtaining pressure distributions within the jet that correspond to experimentally measured distributions. The authors' earlier results for jet flow into a generating bank geometry are briefly reviewed; after this, the extension of that model to an economizer geometry is presented, and finally a new approach to modelling jet development is presented that yields results showing better agreement with the authors' experimental results.

EARLIER RESULTS

Figure 1 illustrates the authors' first [6] use of Fluent to model sootblower jet flow



SHAHED DOROUDI
University of Toronto
Toronto, Ontario
Canada

AMEYA POPHALI
University of Toronto
Toronto, Ontario
Canada

MARKUS BUSSMANN
University of Toronto
Toronto, Ontario
Canada
*Contact: bussmann@mie.utoronto.ca

DANNY TANDRA
Clyde Bergemann Inc.,
Atlanta,
USA

HONGHI TRAN
University of Toronto
Toronto, Ontario
Canada

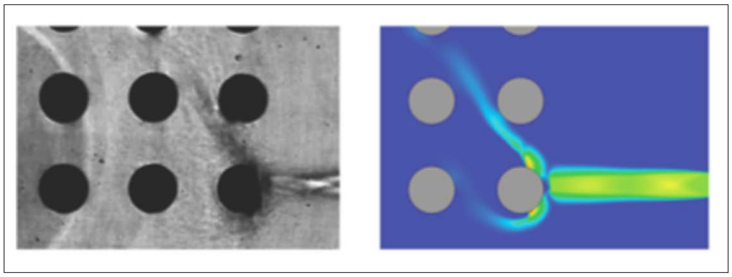


Fig. 1 - Comparison of a schlieren image (left) of jet flow into a generating bank with a Mach contour ([6]).

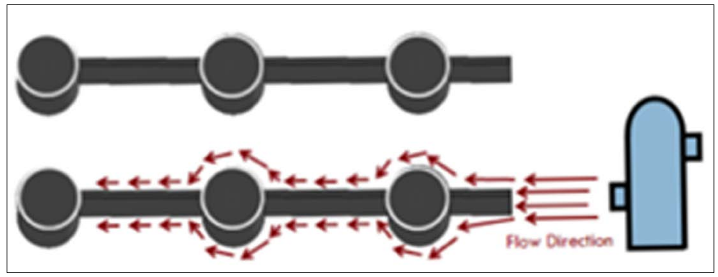


Fig. 2 - Schematic of the finned tube arrangement in an economizer.

into a generating bank geometry. The schlieren image on the left represents a slightly under-expanded (pressure ratio~1.3) Mach 2.5 air jet. The image on the right is a Mach contour of a Fluent simulation in which the ANSYS ICEM CFD commercial software was used to generate the 2.5 million cell three-dimensional tetrahedral mesh on a rectangular domain, extending from the sootblower nozzle exit into an array of cylinder tubes. The known air mass flow rate (from the schlieren experiments) was imposed as an inlet boundary condition at the nozzle outlet; the pressure at all other boundaries of the domain was specified as zero gauge pressure. Figure 1 illustrates that the resulting Mach contour (along the mid-plane of the jet) is in good qualitative agreement with the schlieren image. However, the corresponding pressure contour along the jet core centerline fails to show the pressure fluctuations that correspond to the Mach number fluctuations apparent in Fig. 1. Instead, the simulation predicts that the jet core essentially maintains the nozzle exit pressure of approximately 2 MPa up to the point of impingement with the first tube. This pressure field behaviour is not consistent with the experimental data, and the lack of any pressure loss along the jet centerline before impingement suggests that the inlet boundary condition was not specified correctly.

INITIAL ECONOMIZER MODEL

The results of the generating bank model, despite qualitative agreement, prompted the authors to revisit the model, and in particular the effect of the mesh and boundary conditions on the results. This effort was focussed on modelling jet flow into an economizer section for which schlieren and pitot tube results are also available [7]. As illustrated in Fig. 2, the economizer has a very complex geometry, with an array of cylindrical tubes connected by flat plate fins.

Figure 3 illustrates the economizer model and the boundary conditions imposed. Two key dimensions of the experiment, the tube diameter (Dt) and the nozzle exit diameter (De), were used to scale the domain dimensions. A mass flow rate of 41.5 g/s was imposed as an inlet condition at the circular nozzle exit (De = 7.4 mm). This calculated flow rate corresponded to an inlet Mach number of 2.5. The entire domain was initialized to a temperature of 20°C to match the room temperature condition of the Pophali experiments [7]. Gas density was calculated from temperature and pressure using the ideal gas law, and therefore the calculation included solution of the energy equation.

For all the Fluent simulations, the standard k-ε turbulence model was used along with two user-defined functions (UDFs). The first UDF applied Heinz's [8] structural compressibility correction to reduce the turbulent kinetic energy (the k term) redistribution. The second UDF disabled an unnecessary pressure-dilatation correlation term based on the work of Sarkar [9], which is implemented by default in ANSYS Fluent version 14.5.

An initial mesh was created by dividing the domain into small quadrilateral volumes which could then be meshed independently. This made it possible, for example, to produce a fine mesh down-

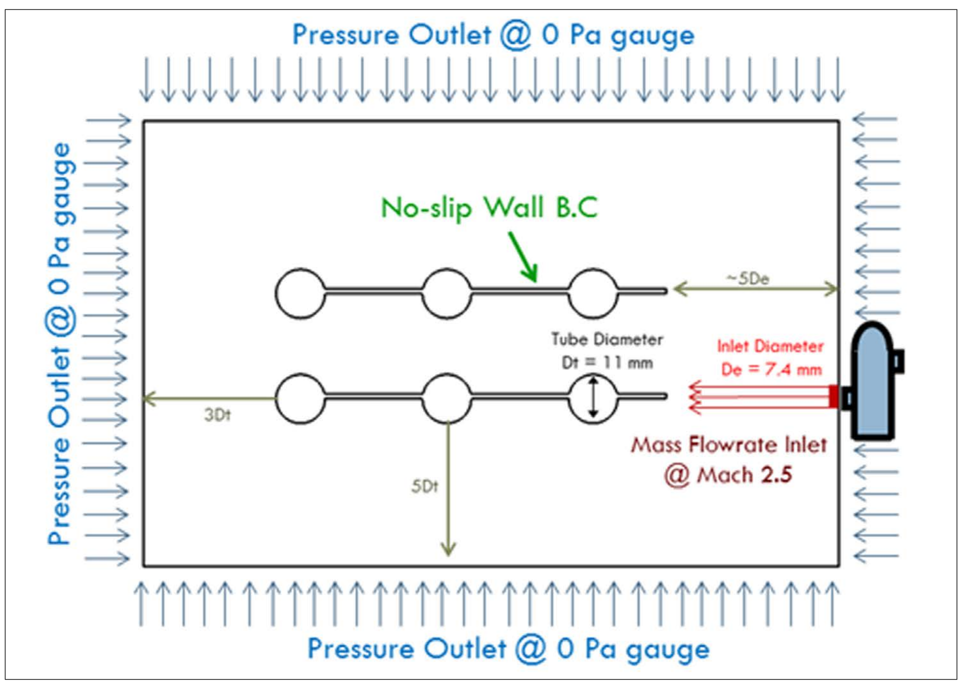


Fig. 3 - The economizer model: boundary and inlet conditions.

stream of the nozzle exit, while leaving volumes with little flow relatively coarse, thus reducing the computation time. A hybrid mesh of 1.8 million tetrahedral and hexahedral elements was created, but converged results could not be obtained due to its poor overall quality. Mesh quality is based on two main factors: orthogonality and smoothness. Orthogonality, in the case of a 3D mesh, is a measure of how closely an average element resembles a cuboid. Smoothness is a measure of how well mesh size transitions across the domain; large size variations between adjacent elements can lead to numerical errors and therefore lower mesh quality.

A much better mesh, shown in Fig. 4, was created by further dividing the geometry into sub-segments, particularly around the finned tubes. A rectangular volume around each tube was sliced into eight smaller quadrilateral sections. In the sliced volumes, the element growth rate along the slice lines was used to control radial mesh growth around each tube. For the case presented in Fig. 4, where the jet impinges directly on the lower tube, the radial mesh growth is refined on this particular tube. Using this novel meshing approach, a high-quality mesh consisting of only hexahedral elements with high orthogonality and smoothness could be created. As shown by the image on the right in Fig. 4, the mesh is refined as it approaches the mid-plane, where interesting flow features are most prominent. The top view of the mesh, on the left in Fig. 4, is extruded into the page in a non-linear manner to create the 3D mesh.

ANSYS Fluent offers a choice of pressure- and density-based solvers. Traditionally, the pressure-based approach has been deemed the solver of choice for subsonic incompressible flows and the density-based approach for handling the shocks and steep gradients characteristic of supersonic compressible flows [10]. However, the ongoing development of Fluent has improved the pressure-based solver to the point that it can be used to resolve shocks in highly compressible flows. Both solvers were tested in this research, and for unknown reasons, the second-order

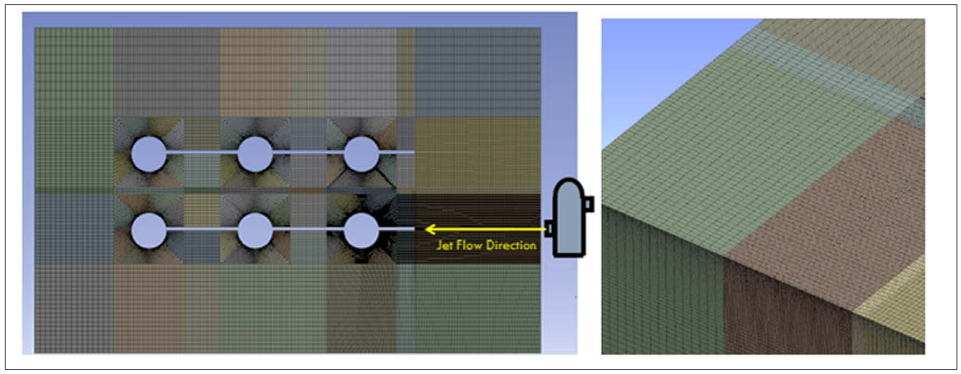


Fig. 4 - **Final economizer mesh.**

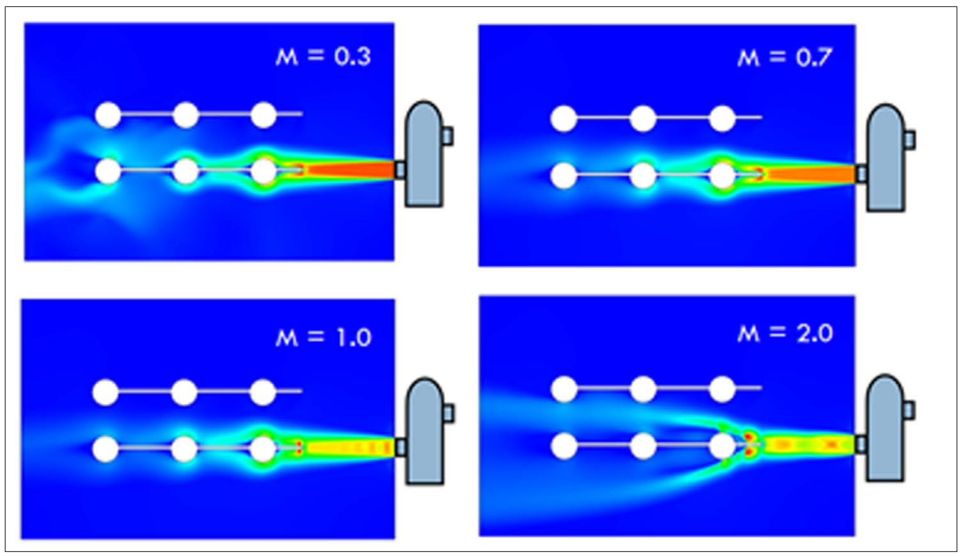


Fig. 5 - **Mach field development from Ma = 0.3 to 2.0.**

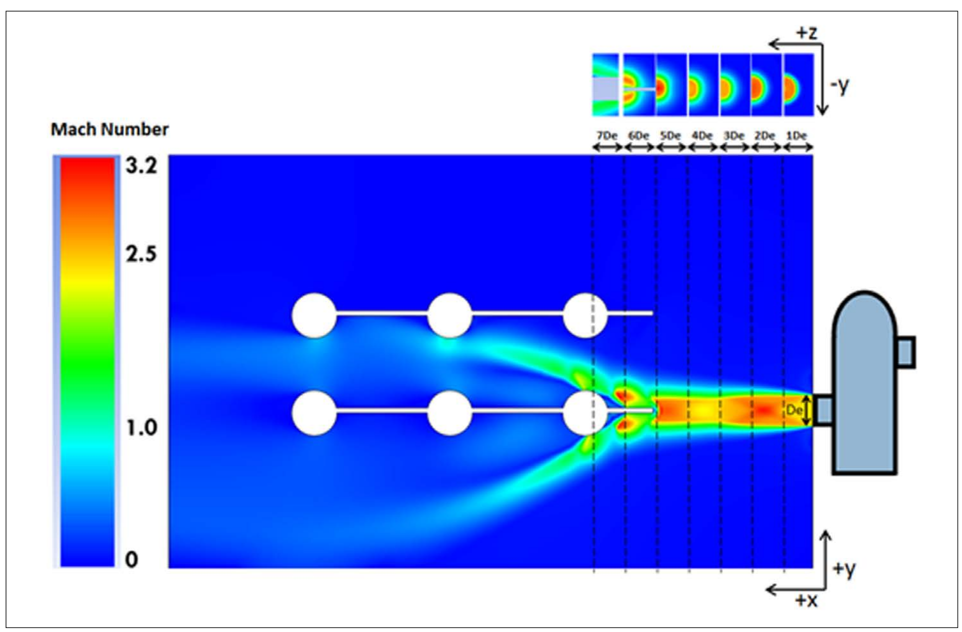


Fig. 6 - **Mach field contour for an inlet Mach number of 2.5.**

pressure-based solver had more success in obtaining converged results.

Appropriate domain initialization is critical to ensure convergence towards a solution. Supersonic flows are somewhat notorious for being difficult to initialize. The Fluent User Guide [11] recommends a strategy of starting the flow with a subsonic Mach number and incrementally increasing Ma to the desired value. As shown in Fig. 5, a first simulation was run at an inlet Mach number of 0.3, following which the value was increased until the desired final result was reached, as presented in Fig. 6. This figure clearly shows how the Mach number fluctuates along the jet core; this behaviour is further illustrated by the seven insets at the top of the figure that illustrate Mach contour cross-sections at each of seven nozzle exit diameters (De) downstream of the nozzle exit and the gradual radial expansion of the jet with distance from the nozzle exit. Figure 7 shows this result again, with two other simulation results, for non-zero offsets (the distance between the jet centerline and the leading fin), all compared to schlieren images of the same region. As with the generating-bank results, the Mach contours show surprisingly good qualitative agreement with the experiments. The 3D nature of the jet and its decay in the vertical direction are illustrated in Fig. 8.

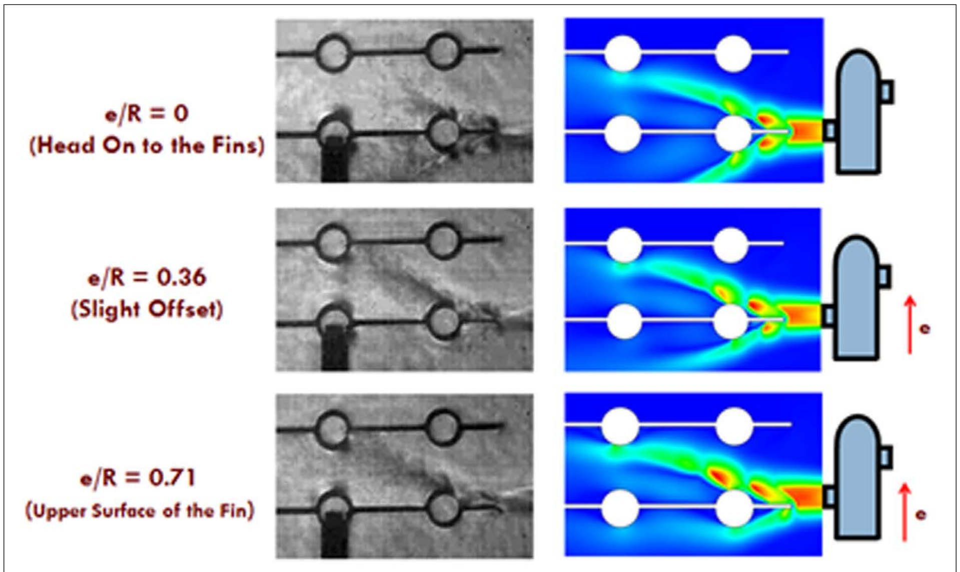


Fig. 7 - Comparison of experimental and simulation results at different offsets.

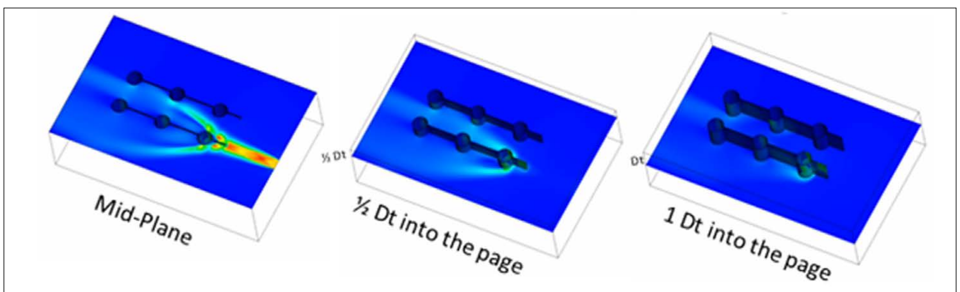


Fig. 8 - For the zero-offset case, Mach contours show how the jet decays with distance into the page. Dt is the tube diameter.

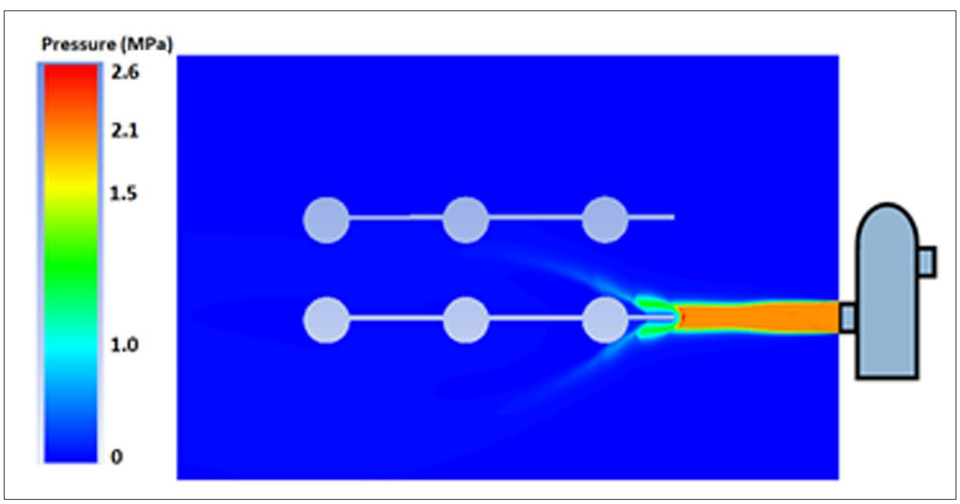


Fig. 9 - Pressure contour of jet flow onto the economizer at zero offset.

Unfortunately, as with the simulations of jet flow into the generating bank, the economizer results also failed to capture the pressure fluctuations associated with an under-expanded jet. Figure 9 illustrates that the pressure along the jet centerline to the point of impingement essentially remained at the nozzle inlet value of 2.14 MPa.

A DIFFERENT APPROACH TO THE INLET BOUNDARY CONDITION

In all previous work on sootblower modelling, the authors simulated the flow from the nozzle exit plane outwards, and their earlier work with CFDLib [4,5] yielded accurate pressure fluctuations downstream of the nozzle exit. So far, it has proved impossible to replicate this with ANSYS Fluent, and so recently a different approach, based on other work [12], has been chosen that uses Fluent to model similar jets.

Under-expansion of a jet results from a pressure discontinuity at the nozzle exit plane. In the case of the Pophali experiments [7], the static pressure infinitesimally upstream of the nozzle exit

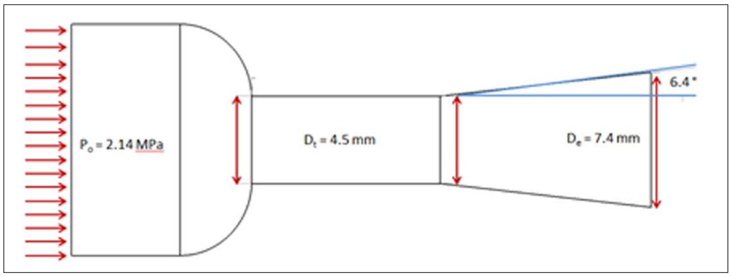


Fig. 10 - Schematic of the nozzle used for the schlieren experiments [7].

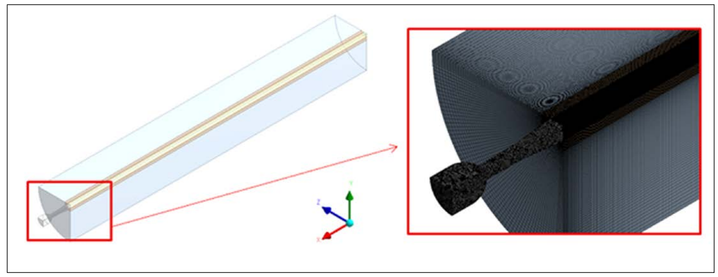


Fig. 11 - One-quarter cylindrical mesh, including the nozzle, for a free jet simulation.

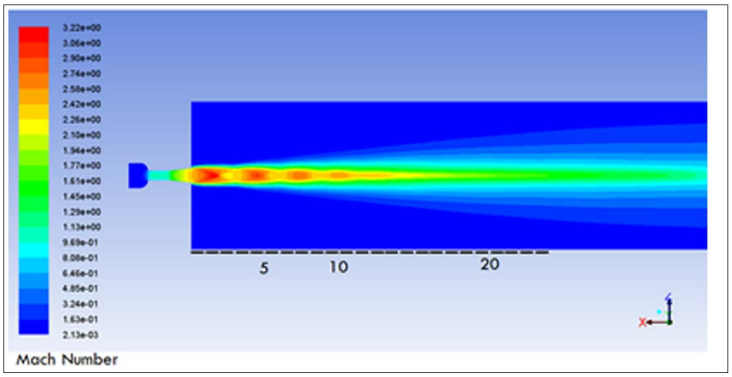


Fig. 12 - Mach contour of the free jet, including the zone within the nozzle. The scale beneath the jet indicates nozzle diameters downstream of the nozzle exit.

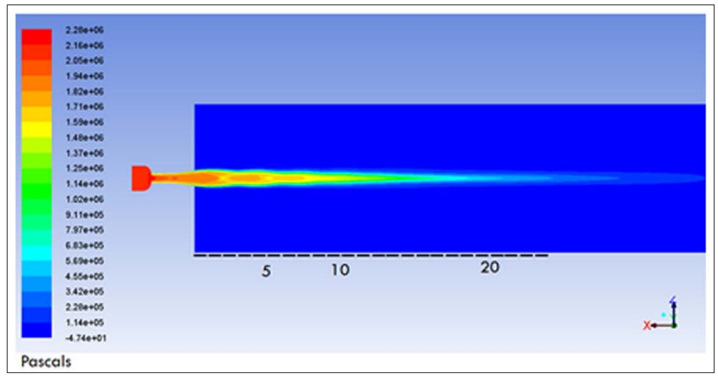


Fig. 13 - Pressure contour of the free jet.

is 30% higher than the ambient pressure infinitesimally downstream of the nozzle. Consequently, perhaps the omission of the nozzle from the computational domain leads to a loss of flow information that Fluent requires to predict pressure fluctuations.

Figure 10 shows the nozzle used by Pophali [7] for the one-quarter scale schlieren visualizations. Figure 11 illustrates a mesh that includes this nozzle. Rather than immediately testing this approach on the economizer model, Fig. 11 illustrates a simple one-quarter cylindrical mesh for simulating an axisymmetric free jet. Instead of specifying a mass flow rate, the known compressed-air tank pressure of 2.14 MPa was specified at the upstream end of the nozzle.

Simulations of this free jet (including the nozzle) successfully converged to second-order accuracy using the implicit density-based solver. (This is the solver that could not be used for the earlier economizer simulations.) A different strategy was also used to initialize the simulations and accelerate convergence. The full multi-grid (FMG) feature in Fluent involves constructing a number of grid levels of varying coarseness. The flow is solved for quickly on the coarsest grid level; that solution is then interpolated onto the next finer level. This process is repeated until a solution is obtained on the finest level, which is the original grid. The default FMG initialization settings of ANSYS Fluent were used to initialize the free jet model. The results, presented in Figs. 12 to 14, show approximately five shock cells and a corresponding fluctuation in the total pressure value.

Figure 14 shows a plot of the centerline pressure of this preliminary free jet, which demonstrates reasonable agreement with experimental trends. The supersonic core of the jet, as measured experimentally, is about 18 nozzle diameters long, and this length is reasonably predicted by the simulation. Furthermore, the pressure decay follows a similar trend. All that remains is to predict the pressure magnitude accurately, beginning at the nozzle exit. The authors are hopeful of soon produc-

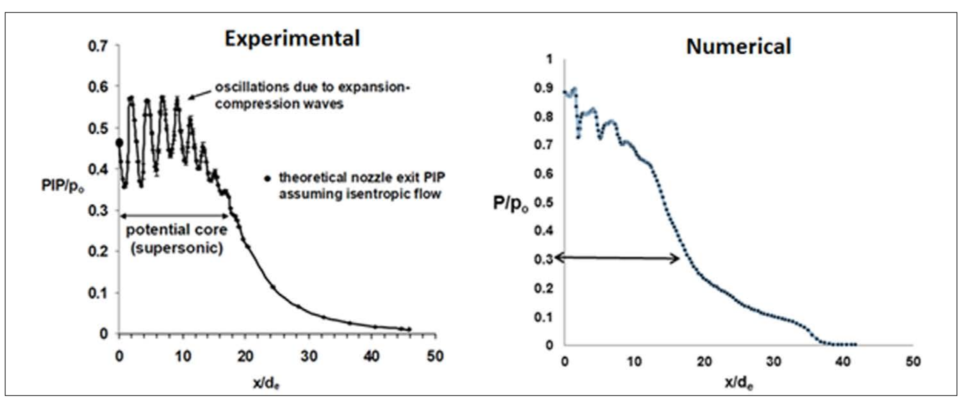


Fig. 14 - Normalized pressure vs distance from the nozzle exit (De = 7.4 mm and Po = 2.14 MPa).

ing simulations that predict these jets at least as accurately as in previous studies [4,5].

FUTURE WORK

The authors have spent considerable time using Fluent to model high-speed sootblower jets and appear close to identifying the appropriate boundary and initial conditions that will yield accurate descriptions of these flows, including pressure fluctuations within off-design jets. Once this initial development work has been completed, future work will focus on using Fluent to model complex jet/tube interactions and, more importantly, the jet/deposit interactions that are of most interest for assessing and improving sootblowing effectiveness.

ACKNOWLEDGEMENTS

This work was conducted as part of the research program on "Increasing Energy and Chemical Recovery Efficiency in the Kraft Process - III", jointly supported by the Natural Sciences and Engineering Research Council of Canada (NSERC) and a consortium of the following companies: Andritz, AV Nackawic, Babcock & Wilcox, Boise, Carter Holt Harvey, Celulose Nipo-Brasileira, Clyde-Bergemann, DMI Peace River Pulp, Eldorado, ERCO Worldwide, Fibria, FP Innovations, International Paper, Irving Pulp & Paper, Kiln Flame Systems, Klabin, MeadWestvaco, Metso Power, StoraEnso Research, Suzano, Tembec, and Tolko Industries.

REFERENCES

1. Pophali, A., Emami, B., Bussmann, M., and Tran, H.N., "Studies on Sootblower Jet Dynamics and Ash Deposit Removal in Industrial Boilers", Fuel Processing Technology, 105, 69−76 (2013).
2. Tandra, D.S., "Development and Application of a Turbulence Model for a Sootblower Jet Propagating Between Recovery Boiler Superheater Platens", Ph.D. Thesis, University of Toronto (2005).
3. Tandra, D.S., Kaliazine, A., Cormack, D.E., and Tran, H.N., "Numerical Simulation of Supersonic Jet Flow Using a Modified k−ε Model", International Journal of Computational Fluid Dynamics, 20, 19−27 (2006).
4. Emami, B., "Numerical Simulation of Kraft Recovery Boiler Sootblower Jets", Ph.D. Thesis, University of Toronto (2009).
5. Emami, B., Bussmann, M., and Tran, H.N., "Application of Realizability and Shock Unsteadiness to k−ε Simulations of Under-Expanded Axisymmetric Supersonic Free Jets". Journal of Fluids Engineering, 132, 041104-1−041104-7 (2010).
6. Bussmann, M., Emami, B., Tandra, D.S., Lee, W.-Y., Pophali, A., and Tran, H.N., "Modelling of Sootblower Jets and the Impact on Deposit Removal in Industrial Boilers", Energy & Fuels, 27, 5733-5737 (2013).
7. Pophali, A., "Interaction between a Supersonic Jet and Tubes in Kraft Recovery Boilers", Ph.D. Thesis, University of Toronto (2011).
8. Heinz, S., "A Model for the Reduction of the Turbulent Energy Redistribution by Compressibility", Physics of Fluids, 15, 3580−3583 (2003).
9. Sarkar, S., "The Pressure Dilatation Correlation in Compressible Flows", Physics of Fluids A, 4, 2674−2682 (1992).
10. Fluent User Guide, version 14.5, "25.1 Overview of Flow Solvers.
11. Fluent User Guide, version 14.5, "9.6.5 Solution Strategies for Compressible Flows.
12. Garcia, R.G., "CFD Simulation of Flow Fields Associated with High-Speed Jet Impingement on Deflectors", M.S. Thesis, Virginia Polytechnic Institute and State University (2007).