MODELING SOOTBLOWER JET EFFECTIVENESS WITH ANSYS FLUENT

Shahed Doroudi, Ameya Pophali, Markus Bussmann, Danny Tandra and Honghi Tran
University of Toronto
Toronto, Ontario, Canada

ABSTRACT

Sootblower jet effectiveness is a strong function of the force exerted by a jet on a fireside deposit, that depends strongly on the local geometry. In the superheater section of a recovery boiler, the spacing between platens is generous and so sootblower jets have easier access to deposits. Tubes in the generating bank and economizer sections are much more closely spaced, and so jet access to deposits on tubes beyond the first row is limited. In either case, the interaction of a supersonic steam jet with tubes and deposits is a complex phenomenon. Research at the University of Toronto over the past decade has examined the dynamics of sootblower jet interaction with tube geometries characteristic of a recovery boiler. The work has involved both experiments and CFD (computational fluid dynamics) analyses. The early CFD studies employed a research code that was difficult to apply to more complex geometries. Recently, turbulence model corrections that were developed during that time have been incorporated into ANSYS Fluent version 14.5, allowing us 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: our attempts to identify an appropriate inlet boundary condition at the sootblower jet nozzle, that will yield 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 up to 10% or more of the steam generated by a boiler, and so represent a significant cost. This motivates research into optimizing sootblowing: 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 modeling, as summarized in a recent overview [1]. The modeling 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 underexpanded) jets impinging on geometries characteristic of superheater platens in recovery boilers. Recently we began to use the commercial CFD software ANSYS Fluent to model jet/tube interactions in more complex geometries like those of the generating bank [6], comparing the results to schlieren images and pitot tube measurements of air jet flow into ¼ scale models of recovery boiler superheater, generating bank and economizer geometries [7].

Here we report on recent work using ANSYS Fluent to model sootblower jet flow into an economizer geometry, and the challenge of obtaining pressure distributions in the jet that correspond to experimentally measured distributions. We briefly review our earlier results of jet flow into a generating bank geometry, then present the extension of that model to an economizer geometry, and finally present a new approach to modeling the development of the jet that yields results that show better agreement with our experimental results.

EARLIER RESULTS

Figure 1 illustrates our first [6] use of Fluent to model sootblower jet flow into a generating bank geometry. The schlieren image on the left is of a slightly underexpanded (pressure ratio ~ 1.3) Mach 2.5 air jet. The image on the right is a Mach contour of a Fluent simulation, where the commercial software ANSYS ICEM CFD was used to generate the 2.5 million cell three-dimensional tetrahedral mesh on a rectangular domain, 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 Figure 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 behavior of the pressure field is not consistent with the experimental data, and the lack of any pressure loss along the jet centerline prior to impingement suggests that we did not specify the inlet boundary condition correctly.

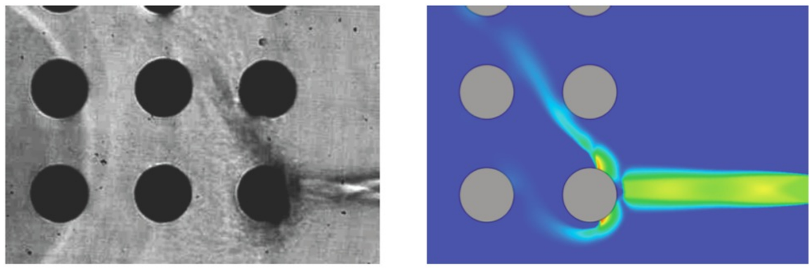

Figure 1. Comparison of a schlieren image (left) of jet flow into a generating bank with a Mach contour (from [6]).

AN INITIAL ECONOMIZER MODEL

The results of the generating bank model, despite qualitative agreement, prompted us to revisit the model, and in particular the role of the mesh and boundary conditions on the results. We focused this effort on modeling jet flow into an economizer section, for which we also have schlieren and pitot tube results [7]. As illustrated in Figure 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 that are imposed. Two key dimensions of the experiment, the tube diameter (Dt) and the nozzle exit diameter (De), are used to scale the domain dimensions. A mass flow rate of 41.5 g/s is imposed as an inlet condition at the circular nozzle exit (De = 7.4 mm). This calculated flow rate corresponds to an inlet Mach number of 2.5. The entire domain is initialized to a temperature of 20^{o}C to match the room temperature condition of the Pophali experiments [7]. Gas density is calculated from temperature and pressure using the ideal gas law, and so the calculation includes a solution of the energy equation.

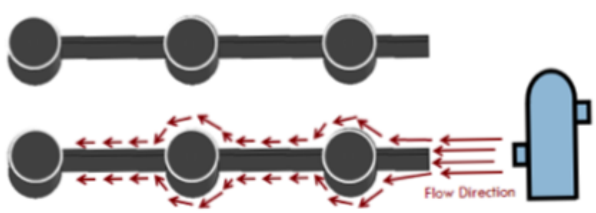

Figure 2. Schematic of the finned tube arrangement of an economizer.

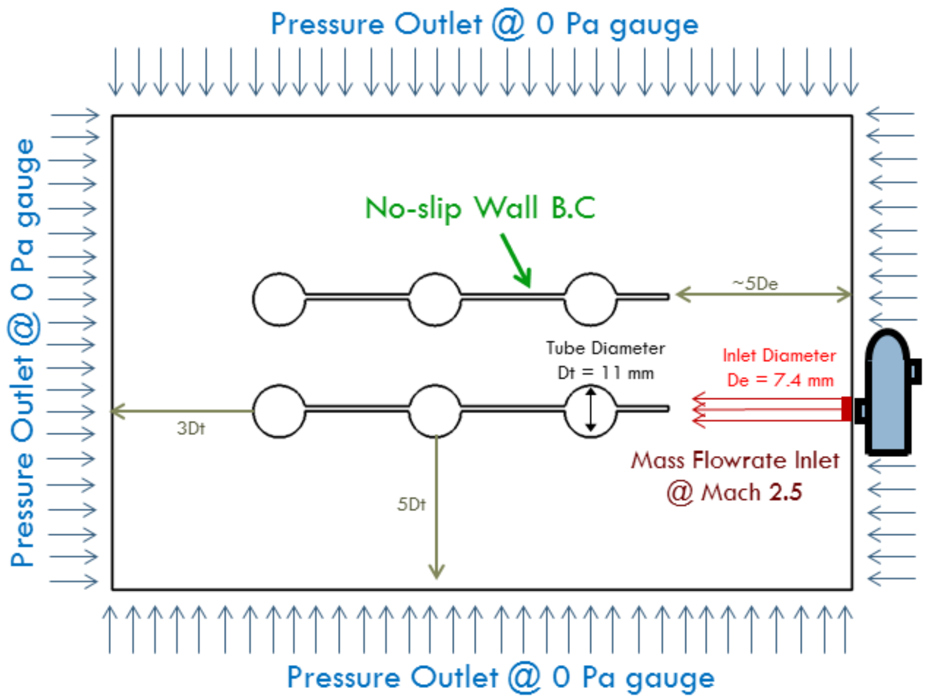

Figure 3. The economizer model - boundary and inlet conditions.

For all of the Fluent simulations, the standard k-ε turbulence model was utilized along with two User Defined Functions (UDFs). The first UDF applies Heinz's [8] structural compressibility correction to reduce the turbulent kinetic energy (k term) redistribution. The second UDF disables 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 allowed us, for example, to produce a fine mesh downstream 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 we were unable to obtain converged results due to its poor overall quality. The quality of a mesh is based mainly on two factors: orthogonality and smoothness. The 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 thus a lower mesh quality.

A much better mesh, shown in Figure 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. Once sliced, the element growth rate along the slice lines was used to control the radial growth of the mesh around each tube. For the case presented in Figure 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, we were able to create a high quality mesh of only hexahedral elements, with high orthogonality and smoothness. As shown by the image on the right in Figure 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 Figure 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. We tested both solvers, and for unknown reasons, had more success obtaining converged results with the second-order pressure-based solver.

Appropriate initialization of the domain is critical for ensuring 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 Figure 5, we ran a first simulation at an inlet Mach number of 0.3, and then increased the value until we reached the desired final result, presented in Figure 6. This figure clearly shows how the Mach number fluctuates along the jet core; this behavior 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 illustrate the gradual radial expansion of the jet with distance from the nozzle exit. Figure 7 shows this result again, and 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. 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, is illustrated in Figure 8.

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

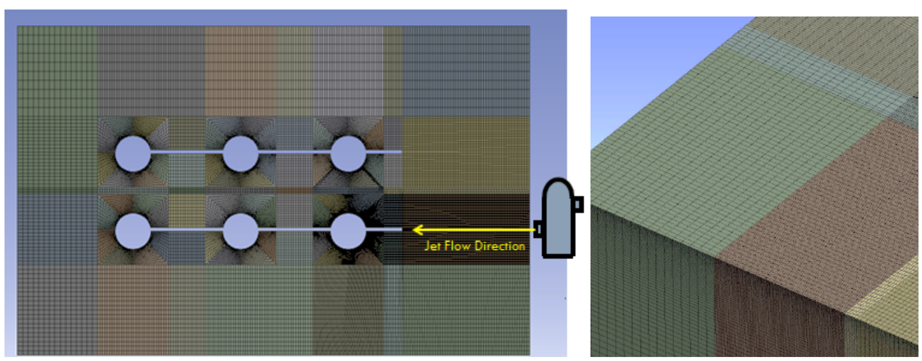

Figure 4. The final economizer mesh.

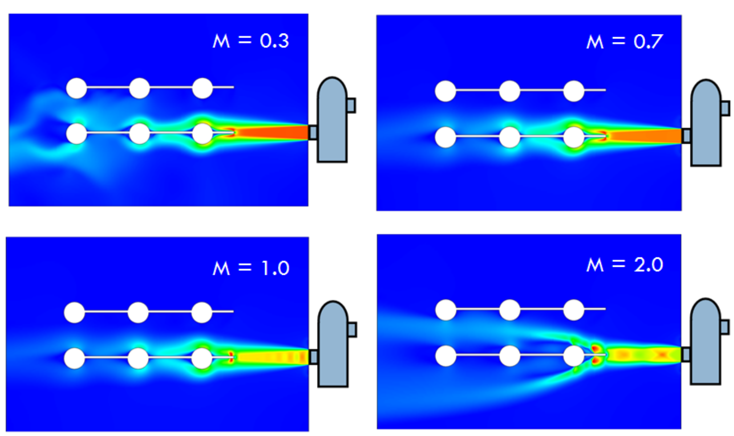

Figure 5. Mach field development from Ma = 0.3 to 2.0.

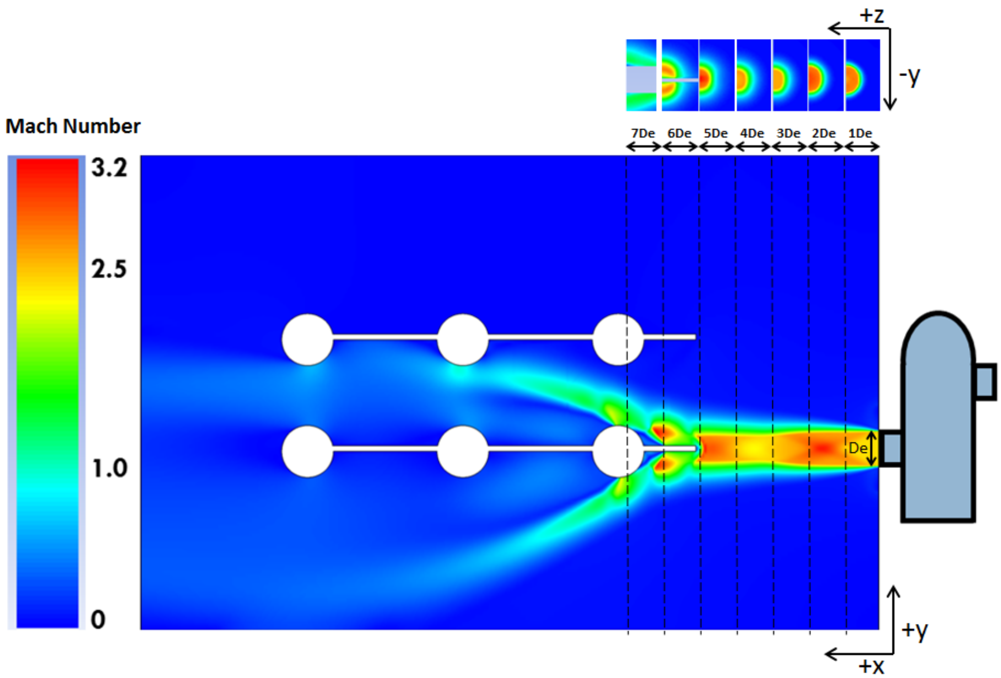

Figure 6. Mach field contour for an inlet Mach number of 2.5.

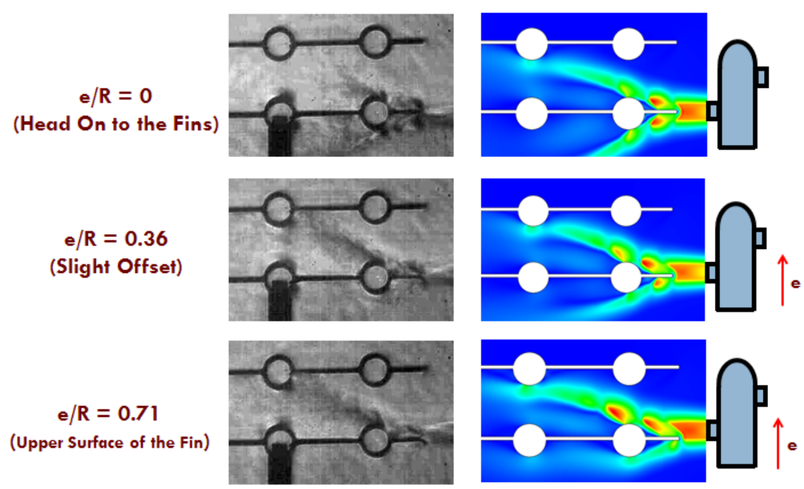

Figure 7. Comparison of experimental and simulation results at different offsets.

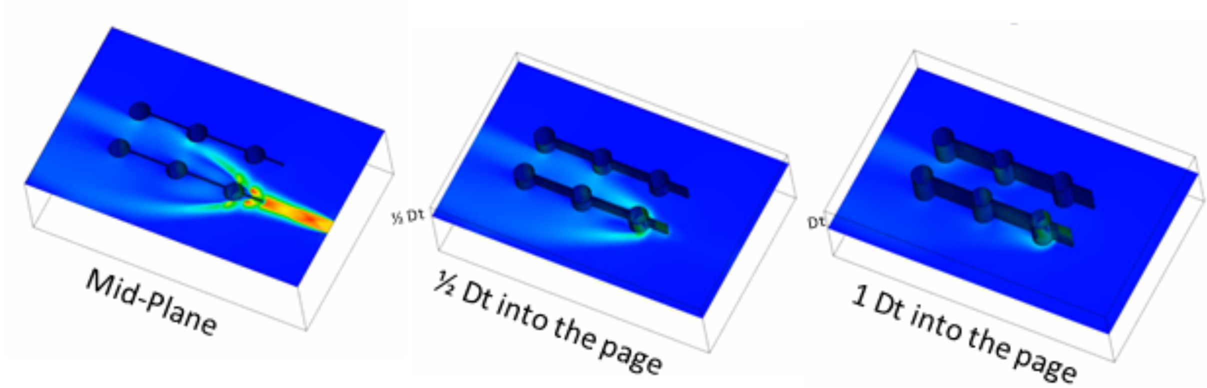

Figure 8. For the zero offset case, Mach contours show how the jet decays with distance into the page. Dt is the tube diameter.

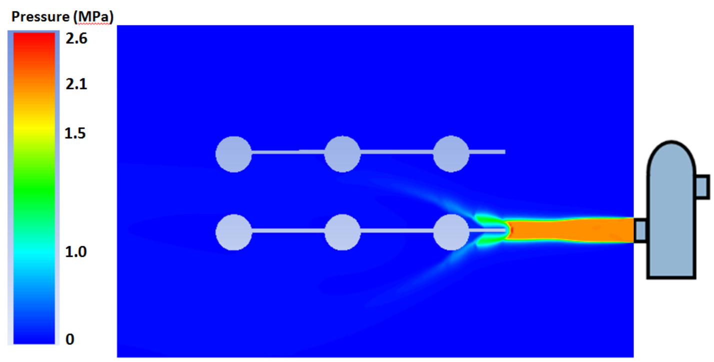

Figure 9. Pressure contour of jet flow onto the economizer at zero offset.

A DIFFERENT APPROACH TO THE INLET BOUNDARY CONDITION

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

The underexpansion of a jet is a phenomenon that 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 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 in order to predict the pressure fluctuations.

Figure 10 shows the nozzle used by Pophali [7] for the ¼ scale schlieren visualizations. Figure 11 illustrates a mesh that includes this nozzle. Rather than immediately test this approach on the economizer model, Figure 11 illustrates a simple quarter cylindrical mesh for the simulation of an axisymmetric free jet. Instead of specifying a mass flow rate, the known compressed air tank pressure of 2.14 MPa is 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 we were not able to use for the earlier economizer simulations.) We also employed a different strategy 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 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 Figures 12 to 14, show approximately five shock cells and a corresponding fluctuation in the total pressure value.

Figure 14 is the plot of the centerline pressure of this preliminary free jet, and 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 accurately predict the magnitude of the pressure, beginning at the nozzle exit. We are hopeful that we will soon produce simulations that predict these jets at least as accurately as we have previously [4,5] done.

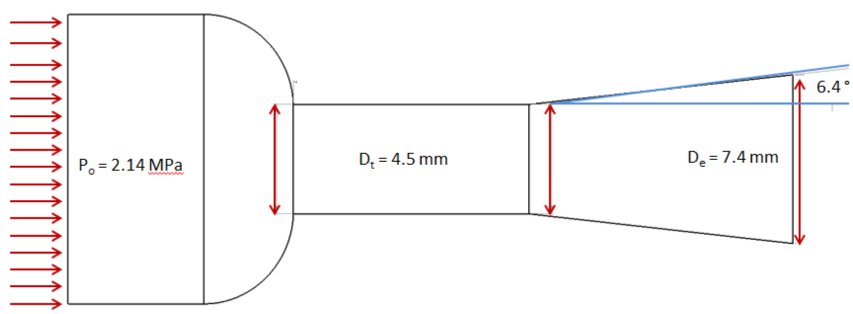

Figure 10. Schematic of the nozzle used for the schlieren experiments [7].

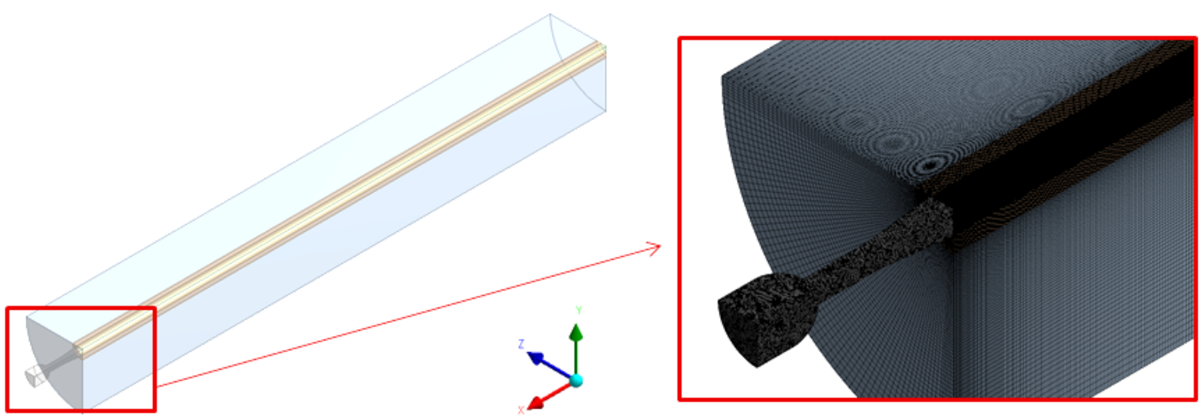

Figure 11. One quarter cylindrical mesh including the nozzle, for a free jet simulation.

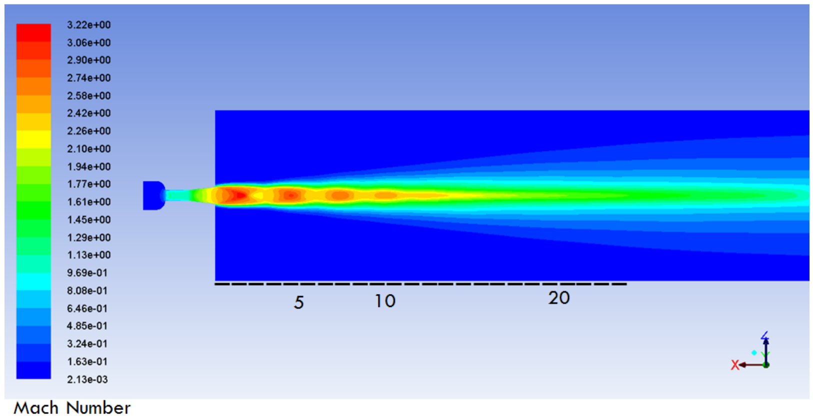

Figure 12. Mach contour of the free jet, including within the nozzle. The scale beneath the jet indicates nozzle diameters downstream of the nozzle exit**.**

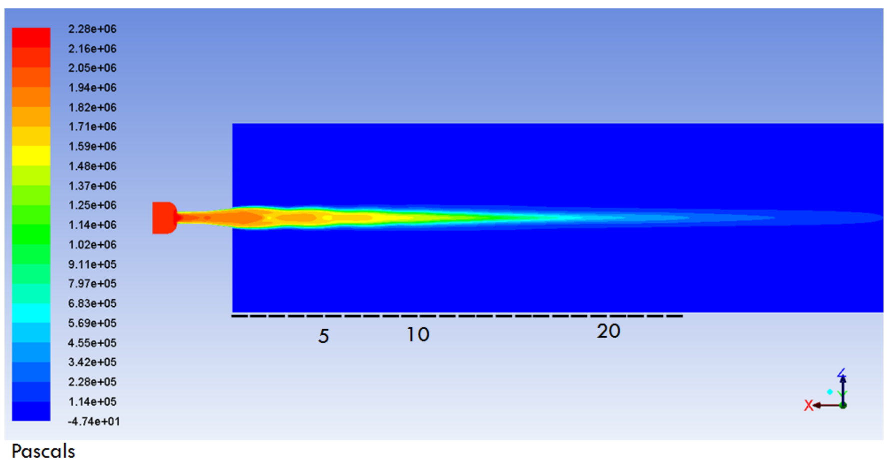

Figure 13. Pressure contour of the free jet.

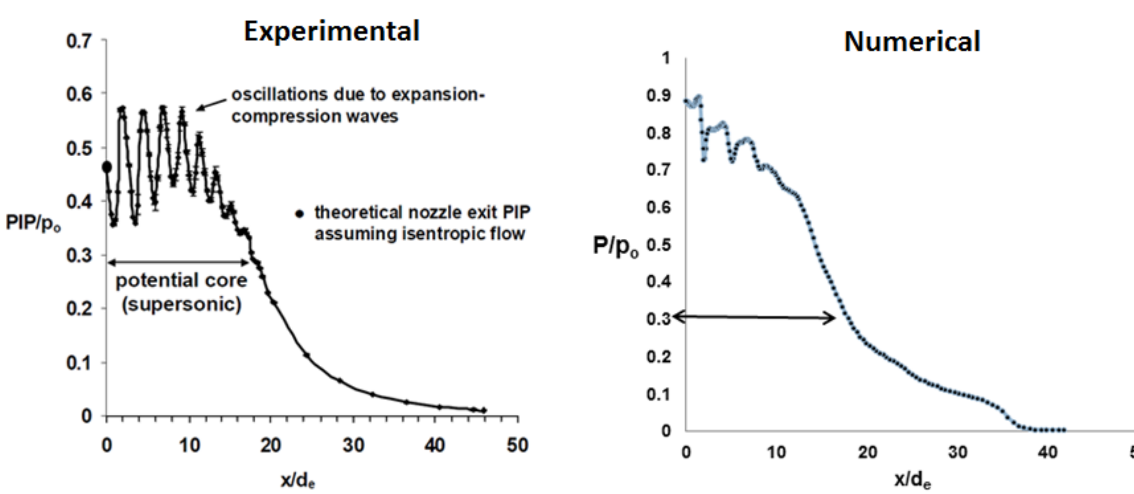

Figure 14. Normalized pressure vs distance from the nozzle exit (de = 7.4 mm and Po = 2.14 MPa).

FUTURE WORK

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

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 2013, 105, 69−76.

[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 2006, 20, 19−27.

[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 2010, 132, 041104-1−041104-7.

[6] Bussmann M., Emami B., Tandra D.S., Lee W.-Y., Pophali A. and Tran H.N. “Modeling of sootblower jets and the impact on deposit removal in industrial boilers,” Energy & Fuels 2013, 27, 5733-5737.

[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 2003, 15, 3580−3583.

[9] Sarkar S. “The pressure dilatation correlation in compressible flows,” Physics of Fluids A 1992, 4, 2674−2682.

[10] Fluent User Guide “25.1 Overview of Flow Solvers”, http://aerojet.engr.ucdavis.edu/fluenthelp/html/ug/node986.htm

[11] Fluent User Guide “9.6.5 Solution Strategies for Compressible Flows”, http://aerojet.engr.ucdavis.edu/fluenthelp/html/ug/node405.htm#sec-compress-solve

[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.