ANSYS Fluent for Modeling Sootblower Jets

Shahed Doroudi[1], Markus Bussmann[1], Danny Tandra[1] and Honghi Tran[2]

[1] *Department of Mechanical & Industrial Engineering*
[2] *Department of Chemical Engineering & Applied Chemistry*
University of Toronto

Email: *shahed.doroudi@mail.utoronto.ca*

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 deposit and tube geometries. 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 examine more complex jet/tube interactions in the generating bank and economizer sections. This paper presents an overview of our latest work on developing those models, and in particular how to specify an 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.

1. INTRODUCTION

Sootblowers operate within a kraft recovery boiler to knock deposits off of heat transfer tubes. A sootblower is a lance tube with two radially opposed nozzles producing two supersonic steam jets capable of exerting enough force to dislodge and erode deposits and keep them from building up. These jets can consume 10% or more of the steam generated by a boiler, and thus represent a significant cost to the operators.

The supersonic nozzles on the lance tube are intended to operate at a certain design pressure, creating a fully expanded jet. However, fluctuations in the supply pressure will cause the nozzle to produce an underexpanded/overexpanded jet. An important characteristic of these "off-design" jets is the formation of multi-cell shock structures in the jet core. Figure 2 illustrates an underexpanded jet adapting to the ambient pressure.
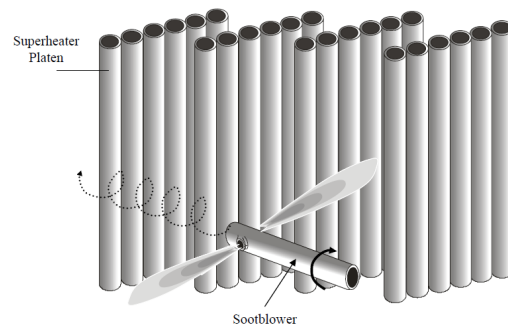
Figure 1. The lance tube rotates in between tube platens; the supersonic sootblower jets attempt to remove accumulated deposits.
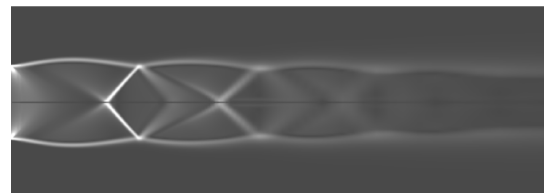
Figure 2. Multi-cell shock structure of an underexpanded jet [12].

The above discussion motivates our research into exploring how to maximize sootblower removal effectiveness while minimizing steam consumption. As summarized in a recent overview [1], research at the University of Toronto in the last decade has examined sootblowing both numerically and experimentally. 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].

In this report, we present an ANSYS Fluent model of a sootblower jet impinging onto the finned tubes of an economizer. We discuss the challenge of obtaining a correct pressure field, which prompted a new approach of including the nozzle in the computational domain. Finally, we present a free jet simulation with this new approach that is in good agreement with experimental results.
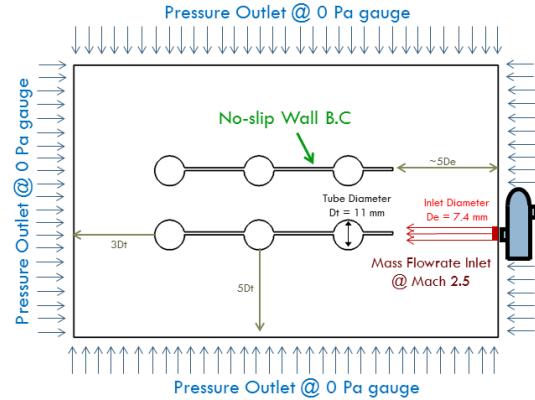
2. ECONOMIZER MODEL

2.1 Boundary Conditions

The economizer section of a recovery boiler consists of cylindrical tubes with interconnected fins. Pophali [1] performed experiments on a ¼ scale model of the economizer. The boundary conditions presented in Figure 3 represent our first ANSYS Fluent model used to simulate the experiments. A rectangular domain of half the geometry (with the top surface corresponding to the jet midplane) was chosen, and the dimensions specified as multiples of the nozzle inlet diameter. A mass flowrate inlet boundary condition was imposed at a value corresponding to an inlet velocity of Mach 2.5. All other exterior surfaces were declared as outlet boundaries at atmospheric pressure (0 gauge).

2.2 Meshing and Preprocessing

The mesh for the economizer model was created using the ANSYS Workbench software. The top surface was sliced into quadrilateral zones to allow for selective refinement. The rectangular volume

around each tube was sliced into eight smaller quadrilateral sections to create an O-grid with a controlled radial growth of elements. The section aligned with the inlet was refined to accommodate for flow gradients within the jet core. The final 3D mesh involved the extrusion, or sweeping (as it's called by ANSYS) of the top surface into the domain, where the cells near the top surface (the mid-plane of the jet) were more refined. The final mesh consisted of 4.75 million hexahedral elements.


Figure 3. An outline of the imposed boundary conditions for the economizer model.
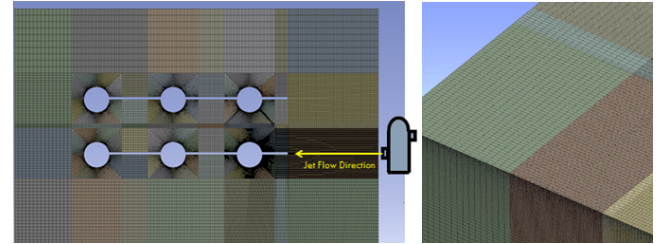
Figure 4. Top view of the mesh (left) and perspective view (right).

The realizable k-ε turbulence model was invoked, along with two UDFs coded specifically for a supersonic underexpanded jet. The first UDF applies Heinz's [8] structural compressibility correction to reduce the turbulent kinetic energy (k) 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 14.5. These UDFs were imposed on the k term via the structural compressibility selection panel, and as a source term addition in ANSYS Fluent.
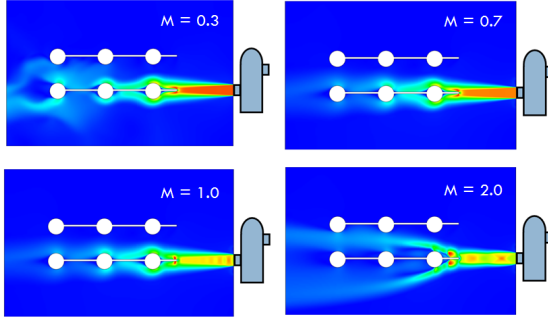
Figure 5. The Mach contours show the development of the flow from incompressible to highly compressible supersonic.

All fluids within the domain were declared as air with density calculated via the ideal gas law. The density-based solver in Fluent never yielded a converged solution. On advice in the Fluent user guide [10], we turned to the pressure-based solver, and initially ran a simulation at a near-incompressible inlet Mach number of 0.3, and then subsequent simulations at higher and higher Mach numbers until we reached Mach 2.5, as illustrated in Figure 5.

2.3 Results

Final results of sootblower jet flow into an economizer are presented in Figures 6 and 7. Although the results appear to predict a good estimate of the Mach field when compared to Pophali's experimental Schlieren visualizations [1], the lack of pressure fluctuations along the jet centerline is not in agreement with the experimental results.
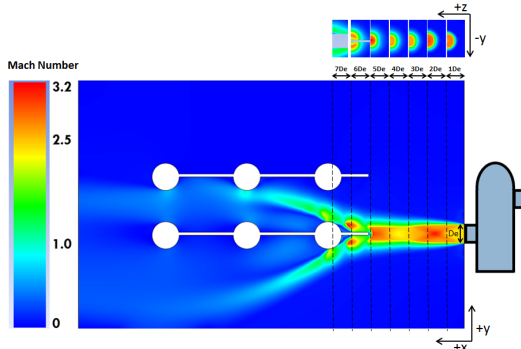
Figure 6. Top view of a Mach 2.5 simulation of a supersonic jet impinging onto an economizer. Cross sections of the jet as a function of the inlet diameter (De = 7.4 mm) are presented on top.
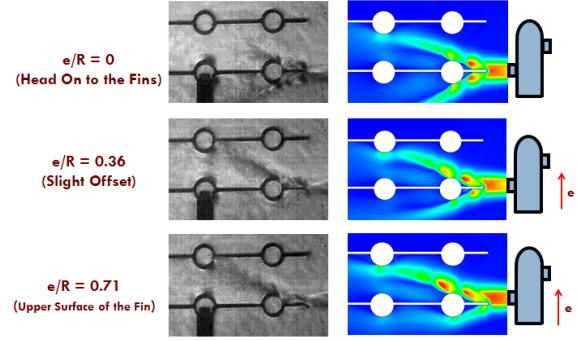
**Figure 7.** The economizer model at various offsets.
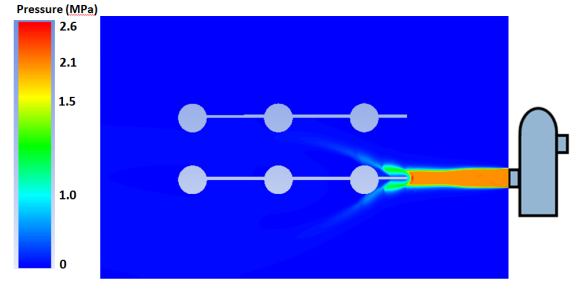
Figure 8. Economizer pressure field shows an almost constant total pressure up to the point of impingement.

3. ADDING THE NOZZLE TO THE DOMAIN

As the first economizer model did not capture the underexpanded nature of Pophali's experimental jet [1], we experimented with adding the nozzle to the computational domain, as has been done elsewhere [12], and initially ran a free jet simulation, for which Pophali also provides data [1].

3.1 Model

For this much simpler geometry, a quarter cylindrical mesh was generated. The Fluent setup was similar to that of the economizer model, except that we returned to the implicit second-order density-based solver, with a new strategy for initializing the simulations to 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. This provides a rough solution as the initial condition on the finest mesh, after which the flow features can be refined through subsequent iterations.
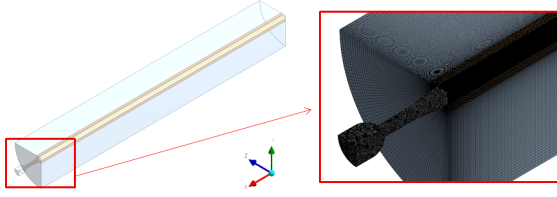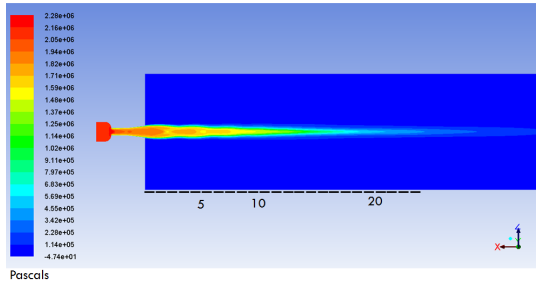
Figure 9. A quarter cylindrical mesh was created with the nozzle included in the computational domain.

3.2   s

Mach Number
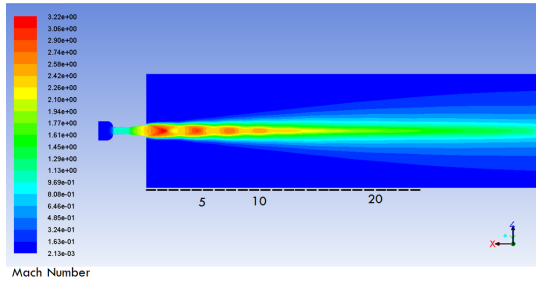
Figure 10. Mach field of the free jet including the nozzle within the computational domain.


Pascals

Figure 11. Pressure field of the free jet including the nozzle within the computational domain.

As seen on Figure 10, as with the economizer simulation, the fluctuations in the Mach field are well predicted. More important, however, is that the pressure field shows fluctuations in the diminishing jet core (Figure 11).

A Peak Impact Pressure (PIP) is an industrial term that represents the localized force that a sootblower can exert on an obstacle and thus is a measure of sootblower effectiveness. Emami [4] defines PIP as: "*the stagnation pressure on the downstream side of a normal shock wave (when the local Mach number is greater than one), or the stagnation pressure itself (in subsonic regions of the flow).*" The Rayleigh equation provides an approximation for the pressure drop across a normal shock at any point in a flow as a function of the local total pressure and Mach number:

$$ PIP = \begin{cases} p_0 & 0 < M \le 1 \\ \\ p_0 \left[\frac{\gamma+1}{2} \frac{M^2}{(1+\frac{\gamma-1}{2}M^2)} \right]^{\frac{\gamma}{\gamma-1}} \left[\frac{2\gamma}{\gamma+1}M^2 - \frac{\gamma-1}{\gamma+1} \right]^{-\frac{1}{\gamma-1}} & M > 1 \end{cases} $$

The variations of centerline total pressure and Mach number from the free jet simulation were used to evaluate an estimate of PIP along the jet core. The figure below shows a comparison of the numerical results to actual pitot tube data [1].
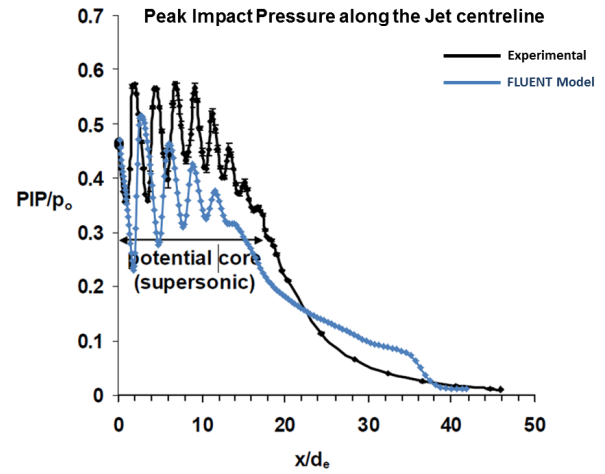
Figure 12. Comparison of predicted PIP estimates with the experimental results by Pophali [1]. PIP values are normalized by the supply pressure P_o = 2.14 MPa; spatial coordinates are normalized by the nozzle exit diameter D_e = 7.4 mm.

4. CONCLUSIONS

It has proven surprisingly difficult to apply ANSYS Fluent to the simulation of compressible jets. Despite great effort, the authors were unable to obtain good predictions of the jet pressure distribution when specifying a mass flowrate boundary condition corresponding to a velocity of Mach 2.5 at the nozzle exit. Adding the nozzle to the domain, and instead specifying the upstream supply pressure and solving for flow through the nozzle yielded much better results. Work is now underway to model jet flow into more complex geometries using this approach.

ACKNOWLEDGEMENTS

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.