MEASURING SOOTBLOWER JET FORCES EXERTED ON DEPOSITS

BY

FADY METTIAS

A THESIS SUBMITTED IN CONFORMITY WITH THE REQUIREMENTS FOR THE DEGREE OF

MASTER OF APPLIED SCIENCE

GRADUATE DEPARTMENT OF MECHANICAL AND INDUSTRIAL ENGINEERING

UNIVERSITY OF TORONTO

© COPYRIGHT BY FADY METTIAS 2019
Measuring Sootblower Jet Forces Exerted On Deposits

Master of Applied Science 2019
Fady Mettias
Department of Mechanical and Industrial Engineering
University of Toronto

ABSTRACT

A computational fluid dynamics (CFD) software package is being used to study the characteristics of a sootblower jet as it interacts with boiler tubes and deposits. With the assumptions being used in the numerical models, the accuracy of these simulations was not previously known. The experimental work performed in this study has quantitatively validated the complex interaction of a supersonic jet interacting with boiler tubes and deposits by experimentally measuring the reaction forces of the jet as well as the pressures imposed on the surface of deposits.

When comparing the experimental and numerical reaction forces, it is evident that the numerical models captured the overall physics of these complex interactions. Similarly, the pressure distributions obtained experimentally by using pressure sensing films on the surface of the deposits agreed well with the numerically predicted surface pressures.
ACKNOWLEDGMENTS

This thesis would not have been possible without the help and support of many individuals, to whom I am very grateful.

First and foremost, I would like to express my sincere gratitude to my supervisors Prof. Mark Kortschot and Prof. Markus Bussmann for their support and guidance throughout my study here at the University of Toronto. Thank you both for your unwavering encouragement and for giving me the opportunity and freedom to grow into a confident researcher. The experiences I gained here were invaluable for my personal and career development.

I would also like to thank Prof. Honghi Tran and Prof. Jan Spelt for being my committee members. Thank you, Prof. Honghi Tran, for your feedback in preparation for our consortium meetings and the support you provided during our update meetings. Furthermore, I would also like to thank Shenglong You for the support he provided with my numerical study.

Additionally, I would like to express my sincere thanks to my lab mates, Joe and Jin, who made my time as a graduate student more enjoyable. Thank you, Ashley, for always being there for me and for your continued support.

Finally, to my parents, Amani and Adel, and to my brother, Shady – thank you for always believing in me. I know I would not be where I am today without all of your support and patience.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>ABSTRACT</th>
<th>ii</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACKNOWLEDGMENTS</td>
<td>iii</td>
</tr>
<tr>
<td>TABLE OF CONTENTS</td>
<td>iv</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>vii</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>viii</td>
</tr>
<tr>
<td>NOMENCLATURE</td>
<td>xii</td>
</tr>
</tbody>
</table>

## 1.0 INTRODUCTION

1.1 Fouling in Recovery Boilers 4

1.2 Sootblowers 5

1.3 Thesis Objectives 7

## 2.0 LITERATURE SURVEY

2.1 Fouling and Sootblowing in Recovery Boilers 8

2.1.1 Deposits in Recovery Boilers 9

2.1.2 Deposit Removal by Sootblowing 13

2.2 Fundamental Hydrodynamics of Sootblowing 16

2.2.1 Nozzle Design and Flow Regime for Sootblower Nozzle 17

2.2.2 Mechanics of Deposit Removal 20

2.3 Previous Experimental Studies 21

2.3.1 In-Situ Measurements of Sootblower Jet 21

2.3.2 Scaled-down Experiments on Brittle Deposits 23

2.3.3 Flow Visualization of Sootblower Jet 24

2.4 Numerical Models of Sootblower Jet 25
LIST OF TABLES

Chapter 2

Table 2.1  Breakup mechanisms of brittle deposits.  24

Chapter 3

Table 3.1  Nozzle and flow details.  31
Table 3.2  Dynamic similarity between the supersonic nozzle and industrial sootblower.  37

Chapter 5

Table 5.1  Comparing the experimentally measured core diameters with the numerically predicted core diameters.  80
LIST OF FIGURES

Chapter 1

Figure 1.1  The Kraft process.  2
Figure 1.2  A typical Kraft recovery boiler.  3
Figure 1.3  Plugging within the Kraft recovery boiler.  4
Figure 1.4  A sootblower removing fireside deposits.  5
Figure 1.5  A long retractable sootblower.  6

Chapter 2

Figure 2.1  The sticky temperature zone of fireside deposits.  9
Figure 2.2  The sintering process.  10
Figure 2.3  The spreading of deposit accumulation over time.  11
Figure 2.4  Deposits forming on the leading edge of the superheater platens.  12
Figure 2.5  Adhesion strength versus boiler tube temperatures in different zones of the recovery boiler.  14
Figure 2.6  Arrangement of superheater platens.  15
Figure 2.7  Comparison of supersonic nozzle with equivalent subsonic nozzle.  17
Figure 2.8  Cross section of converging-diverging nozzle.  17
Figure 2.9  Supersonic jets: (a) fully expanded nozzle; (b) overexpanded nozzle; (c) underexpanded nozzle.  18
Figure 2.10 Development of sootblower nozzle.  19
Figure 2.11 Decay of peak impact pressure for both underexpanded nozzles and fully expanded nozzles.  20
Figure 2.12 Force measurement system.  22
Figure 2.13 Jet force as a function of lance pressure.  22
Figure 2.14 Breakup of model brittle deposits.  23
Figure 2.15 The Schlieren technique.  24
Figure 2.16 Visualization of supersonic jet with superheater platens.  25
Figure 2.17 Comparison of Schlieren images and numerical results of sootblower jet interacting with lab-scale generating bank geometry.  26
Figure 2.18 Comparison of Schlieren images and numerical results of sootblower jet interacting with lab-scale economizer geometry.

Figure 2.19 Static FEA model.

Figure 2.20 FDEM model at time 104 μs.

Figure 2.21 Normalized pressure along centerline axis of supersonic jet.

Figure 2.22 Results obtained from in-situ measurement device.

Figure 2.23 Comparing the gross features of the Schlieren images with the numerical models.

Chapter 3

Figure 3.1 Machined converging-diverging nozzle representing sootblower.

Figure 3.2 Experimental design.

Figure 3.3 Close-up of experimental design.

Figure 3.4 Drying system.

Figure 3.5 Deposit geometries.

Figure 3.6 Typical 3D printed deposit.

Figure 3.7 LabVIEW control.

Figure 3.8 Sensor calibration method.

Figure 3.9 Typical sensor recording.

Figure 3.10 Computational domain and boundary conditions.

Figure 3.11 Typical mesh design.

Comparing experimental versus numerical peak impact pressures: PIP values are normalized by the supply pressure \( p_0 = 2.14 \) MPa; spatial coordinates are normalized by the nozzle exit diameter \( d_e = 7.4 \) mm.

Figure 3.12

Chapter 4

Figure 4.1 Sootblower jet interacting with deposit.

Figure 4.2 Drag force of jet interacting with boiler tube.

Figure 4.3 The decay of a supersonic jet.

Figure 4.4 Final mesh design of jet interacting with boiler tube.
Figure 4.5   Velocity contour of jet interacting with boiler tube.  
Figure 4.6   Drag force of jet interacting with symmetrical deposit.  
Figure 4.7   Velocity contour of jet interacting with symmetrical deposit.  
Figure 4.8   Experimental method for testing smooth deposit.  
Figure 4.9   Simulating rough deposit.  
Figure 4.10  Defining rotation radius of lance.  
Figure 4.11  Angle sootblower rotation.  
Figure 4.12  Trigonometrically relating rotating sootblower to fixed experimental nozzle.  
Figure 4.13  Measured rotation and perpendicular offsets.  
Figure 4.14  Drag force of jet interacting with an asymmetrical deposit at a non-rotated deposit position.  
Figure 4.15  The Coanda effect.  
Figure 4.16  Moment of jet interacting with an asymmetrical deposit at a non-rotated deposit position.  
Figure 4.17  Magnitude and direction of moment.  
Figure 4.18  Drag force of jet interacting with an asymmetrical deposit at a 45° rotated deposit position.  
Figure 4.19  Moment of jet interacting with an asymmetrical deposit at a 45° rotated deposit position.  
Figure 4.20  Comparison between experimental and numerical results for a non-rotated nozzle position.  
Figure 4.21  Velocity contours of jet interacting with an asymmetrical deposit for non-rotated nozzle position at a 7.13 mm offset; (a) 15 mm distance from surface of deposit; (b) 44 mm distance from surface of deposit.  
Figure 4.22  Percent difference between experimental and numerical results.  
Figure 4.23  3D print of true zero position device.  
Figure 4.24  Modeling flexible hose connection; (a) experimental image; (b) modeled fluid domain; (c) mesh design of pipe bend; (d) velocity contour of nozzle domain.  
Figure 4.25  Modeling spherical imperfections; (a) fluid domain; (b) mesh design of imperfections; (c) velocity contour of simulation result.  
Figure 4.26  Modeling cubical imperfection; (a) fluid domain; (b) mesh design of imperfections; (c) velocity contour of simulation result.
## Chapter 5

| Figure 5.1 | Pressure sensing film. | 72 |
| Figure 5.2 | Cross sectional view of Fujifilm Prescale. | 72 |
| Figure 5.3 | Relating film color density to pressure. | 73 |
| Figure 5.4 | Estimating the projected area of the jet. | 73 |
| Figure 5.5 | Fujifilm calibration method. | 75 |
| Figure 5.6 | Calibration stains. | 75 |
| Figure 5.7 | Ultra-low pressure sensing film calibration curve. | 76 |
| Figure 5.8 | Converting grey scale image to pressure contour. | 77 |
| Figure 5.9 | Calculating wrapping pressure. | 78 |
| Figure 5.10 | Comparison of experimental surface pressures and numerical surface pressures; (a) 22 mm from surface of deposit; (b) 44 mm from surface of deposit; (c) 110 mm from surface of deposit. | 79 |
# NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D$</td>
<td>Distance from the center axis of the normal sensor (Figure C.1)</td>
</tr>
<tr>
<td>$d$</td>
<td>Tube diameter</td>
</tr>
<tr>
<td>$D, d_e$</td>
<td>Nozzle exit diameter</td>
</tr>
<tr>
<td>$d_o$</td>
<td>Tube sector angle covered with deposit</td>
</tr>
<tr>
<td>$D_p$</td>
<td>Distance from centerline axis of nozzle to centerline axis of rotary shaft</td>
</tr>
<tr>
<td>$D_t$</td>
<td>Inside diameter of steel ring</td>
</tr>
<tr>
<td>$d_t$</td>
<td>Nozzle throat diameter</td>
</tr>
<tr>
<td>$F$</td>
<td>Net force applied to the deposit in transverse direction</td>
</tr>
<tr>
<td>$F_{\text{applied}}$</td>
<td>Force applied by calibrated weights</td>
</tr>
<tr>
<td>$F_D$</td>
<td>Drag force</td>
</tr>
<tr>
<td>$F_L$</td>
<td>Lift force</td>
</tr>
<tr>
<td>$F_{\text{wrap}}$</td>
<td>Force required to wrap the pressure sensing film</td>
</tr>
<tr>
<td>$H$</td>
<td>Deposit thickness</td>
</tr>
<tr>
<td>$L$</td>
<td>Deposit length measured along the tube axis</td>
</tr>
<tr>
<td>$L_{\text{film}}$</td>
<td>Length of pressure sensing film</td>
</tr>
<tr>
<td>$M$</td>
<td>Net moment of force about the axis of the tube</td>
</tr>
<tr>
<td>$M_{ae}$</td>
<td>Nozzle exit Mach number</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Mass flow rate through nozzle</td>
</tr>
<tr>
<td>$P_\infty$</td>
<td>Ambient pressure</td>
</tr>
<tr>
<td>$P_e$</td>
<td>Nozzle exit pressure</td>
</tr>
<tr>
<td>$P_o$</td>
<td>Nozzle inlet pressure</td>
</tr>
<tr>
<td>$P_{\text{wrap}}$</td>
<td>Pressure exerted to wrap pressure sensing film</td>
</tr>
<tr>
<td>$q_{\text{cond}}$</td>
<td>Condensation produced by Makita compressor</td>
</tr>
<tr>
<td>$q_{\text{in}}$</td>
<td>Inlet air to the Makita compressor</td>
</tr>
<tr>
<td>$q_{\text{out}}$</td>
<td>Outlet air to the Makita compressor</td>
</tr>
<tr>
<td>$R$</td>
<td>Tube radius</td>
</tr>
</tbody>
</table>
\[ r_{deposit} \] Deposit radius
\[ R_e \] Reynolds number
\[ R_{gas} \] Specific gas constant
\[ R.H. \] Relative humidity
\[ R_L \] Lance radius
\[ r_{rotary} \] Radius of rotary shaft
\[ T_o \] Temperature of inlet to nozzle
\[ T_{RD} \] Radical deformation temperature
\[ T_{SK} \] Sticky temperature
\[ v_{da} \] Specific volume of dry air
\[ W_{gr} \] Grain of water per pound of dry air
\[ x \] Distance from center of normal sensors to \( F_{applied} \) (Figure C.1)
\[ x \] Distance from nozzle exit to centerline axis of rotary shaft
\[ Z \] Distance from both the centerline axis of nozzle to centerline axis of rotary shaft

**Greek Symbols**

\[ \sigma_{adh} \] Adhesion strength
\[ \sigma_{max} \] Maximum stress of deposit-tube interface
\[ \sigma_t \] Deposit tensile strength
\[ \phi \] Nozzle rotating angle
\[ \gamma \] Specific heat ratio of gas

**Abbreviations**

\[ PIP \] Peak impact pressure
CHAPTER 1

INTRODUCTION

Although the first few years of the millennium were prosperous for the pulp and paper industry, the newsprint sector saw a decline as the population migrated to digital media. Despite this decline, most of the firms have survived and diversified into paper grades other than newsprint such as household, sanitary papers, and packaging for consumer goods [1]. Since the production of paper products is a very energy intensive process, a need exists to reduce its energy consumption. In 2005, the production of paper products required 6.4 EJ of energy accounting for roughly 6% of the world’s total industrial energy consumption [2].

Paper is produced from raw wood, which is made of cellulose fibers that are bound together by lignin, a binding agent. When the lignin is removed and separated from the cellulose, a pulp mixture known as wood pulp remains and is used to manufacture paper. Wood pulp can be produced in several ways with the most common method being chemical pulping, also referred to
as the Kraft process (Figure 1.1 [3]) [4]. In the Kraft process, wood is cooked in the digester with sodium hydroxide (NaOH) and sodium sulphide (Na₂S) at high pressure and temperature to produce wood pulp. During this process, about half of the wood is dissolved, and forms a mixture of an organic and inorganic chemical called black liquor that is concentrated in evaporators, and then sent to the recovery boiler [3].

One of the most important components in the recovery cycle is the recovery boiler. The recovery boiler is part of a large process that recovers inorganic pulping chemicals, as well as generate steam for the mill (Figure 1.2 [3]). The recovery boiler consists of a furnace bed which is at the bottom of the boiler and three convective heat transfer sections referred to as the superheater, generating bank, and the economizer at the top of the boiler. Feed water flows through the banks of the convective sections in a countercurrent sequence, from the economizer section to the superheater section. As black liquor is sprayed into the lower part of the recovery boiler, it burns producing hot flue gases that travel upwards from the furnace bed and between the tube sections transferring heat to the feed water. As the feed water is heated, it is converted to superheated steam where a portion is sent to the turbines to generate electricity for the mill [3].

Figure 1.1. The Kraft process.
Depending on the quality of the steam, the Kraft process in a typical mill can generate 25 to 35 MW of electricity by burning 1500 tons per day of black liquor in the recovery boiler. This can translate to the recovery boiler generating 60% of the electricity needs of the mill [3]. However, the recovery boiler is difficult to operate at high efficiency since it has many problems related to fouling of heat transfer tubes and plugging of flue gas passages.

**Figure 1.2.** A typical Kraft recovery boiler.
1.1 Fouling in Recovery Boilers

Fouling within a recovery boiler forms as a result of the inorganic material in the black liquor and is mainly comprised of carryover and fume [5]. Carryover particles are the result of mechanical entrainment of black liquor droplets into the flue gas and are relatively large in size (20 um – 3mm). The liquid phase present in the carryover particles is responsible for the accumulation of sticky deposits, which is a function of the flue gas temperature. Fume particles, on the other hand, form when vapors of sodium or potassium compounds in the flue gas condense, and are much smaller than carryover particles (0.1 um – 1um) [6].

In general, the accumulation of deposits on boiler tubes restricts the heat transfer from the hot flue gases to the feed water, which lowers the thermal efficiency of the boiler. If the buildup of these deposits is not well controlled, they will grow in time until they completely block the flue gas passages (Figure 1.3 [7]). When deposit buildup becomes too severe, the massive deposits must be removed by means of water washing, which is very costly to production since it requires the boiler to be taken off-line.
1.2 Sootblowers

Since recovery boiler fouling is a costly and persistent problem for pulp and paper mills, a portion of the steam generated by the boiler is supplied to supersonic nozzles, commonly referred to as a sootblower, in order to help remove deposits as they are formed (Figure 1.4 [8]) [9].

![Diagram of sootblower](image)

**Figure 1.4.** A sootblower removing fireside deposits.

A sootblower consists of a long, hollow steel tube called the lance, with supersonic nozzles at its working end known as converging-diverging nozzles or de Laval nozzles (Figure 1.5 [10]). The sootblower lance periodically translates and rotates into and out of the boiler producing steam jets that have a velocity greater than the speed of sound, which impinge on deposits in order to knock them off. Since there is varying deposit accumulation depending on boiler location, recovery boilers can have up to 100 sootblowers that are operated continuously, each consuming roughly 3-12% of the costly high-pressure steam generated by the boiler [9]. Therefore, optimizing sootblowing to minimize steam consumption while maximizing deposit removal is critical.
The efficiency of the sootblower in removing deposits depends on the characteristics of both the jet and the deposit. Conventionally, the performance of the jet in removing deposits has been correlated to its peak impact pressure (PIP), which is the stagnation pressure of the jet measured along the nozzle centerline downstream of the nozzle exit [7]. During operation, a sootblower jet propagates between different tube arrangements and obstacles creating complicated jet structures that can have adverse effects on its overall cleaning potential. The hostile environment inside the recovery boiler makes it difficult to install sensors or even visualize jet effectiveness. Therefore, much of the experimental work has been conducted using lab-scale experiments to visualize the action of supersonic jets and the breakup of model brittle deposits.

**Figure 1.5.** A long retractable sootblower.
1.3 Thesis Objectives

In order to better understand the mechanisms of deposit removal, finite element solvers are being used to simulate the stress distribution within deposits [11]. These numerical models require direct input from CFD solvers [12, 13], which provide the boundary conditions for the finite element solvers to predict deposit failure. Therefore, the ability of the CFD models to accurately predict the fluid mechanics of sootblower interactions is of great importance. However, as is the case with any CFD analysis, simplifications are made to the models to reduce computational time and complexity. As a result, the accuracy of the simulations may be limited.

A common method of validating CFD outputs is to perform an experimental study to compare its findings with the simulation results. The experimental studies that have been conducted in the past are mainly qualitative and provide only visual validation of the gross features of the CFD analysis. Therefore, the focus of this thesis is on quantitively validating the CFD outputs using lab-scale experimentation.
CHAPTER 2

LITERATURE SURVEY

This chapter reviews studies of deposit formation inside of a recovery boiler, and of sootblower optimization. Since sootblowers use supersonic nozzles, the basic flow principles of supersonic jets are reviewed, and the mechanics of deposit removal are presented. Furthermore, the experimental and numerical studies central to the understanding of the jet and its interaction with boiler tubes and deposits are also presented. Finally, conclusions are drawn from the literature survey.

2.1 Fouling and Sootblowing in Recovery Boilers

The efficiency of the sootblower in removing deposits depends on both the characteristics of the jet and the deposit. The parameters important to the characterization of the jet include the mass flow rate of the steam, the jet decay characteristics, and the distance between the nozzle and the deposit. At the same time, the characteristics of the deposit also influences the sootblower
efficiency. The deposit may be brittle or ductile depending on the location within the boiler, and the removal mechanisms would be different in each case [14].

2.1.1 Deposits in Recovery Boilers

Recovery boiler deposits are comprised principally of sodium sulphate, carbonate, and chloride with smaller concentrations of potassium. They are derived primarily from two distinct sources: carryover and fume. Carryover refers to the relatively large molten or partially-molten smelt-like particles, that impact inertially on tube surfaces; and the deposits are typically hard and tenacious. Fume, on the other hand, is formed as a result of condensation of vapors of sodium and potassium compounds in the flue gas and tends to be soft and powdery. Deposit strength varies from location to location within a recovery boiler because of the differences in formation mechanisms and their melting behavior [7].

Recovery boiler deposits have been shown to have two distinct melting temperatures. The first melting temperature is the temperature at which deposits first begin to melt, and the final

![Figure 2.1. The sticky temperature zone of fireside deposits.](image)
melting temperature is the temperature at which the deposit is completely molten. Between these two temperature extremes, there are two other important temperatures. The first is the sticky temperature, $T_{STK}$, which lies above the first melting temperature and occurs when the deposit contains 15 to 20% liquid phase. The second is the radical deformation temperature, $T_{RD}$, which lies below the final melting temperature and occurs when the deposit contains 70% liquid phase. These critical temperatures define what is known as the sticky temperature zone (refer to Figure 2.1 [7]). When deposits fall in this region, they contain enough liquid phase to stick on tube surfaces and cause deposit accumulation. Outside of this zone, deposition does not occur readily since deposits either do not have enough liquid phase to be sticky or are too fluid and can run off due to their own weight [7]. In addition to this, a diffusional process, known as sintering, will occur at temperatures below the first melting temperature of fume deposits [15]. During sintering, particles bond and the touching surfaces between particles are replaced by grain boundaries to reduce the total surface energy. These points of contact are eventually replaced to form “bridges” between particles (Figure 2.2 [15]). Laboratory and field tests have shown that sintering starts at temperatures about 300 °C (570 °F) and deposits become very hard in less than an hour if exposed to temperatures above 500 °C (930 °F). Their compressive strength has been found comparable to cement making them hard and tenacious, and difficult for sootblowers to remove [7].
The following section describes the formation mechanisms within the three convective regions of the recovery boiler.

**Superheater:** In the lower superheater region, near the bullnose, the flue gas temperature is usually higher than 820 °C (1510 °F). Therefore, carryover deposits are the dominant form of deposition as molten droplets strike and solidify on the tubes to form fused and hard deposits. As the deposits grow thicker, the outer surface temperature increases until it reaches the radical deformation temperature, at which point it becomes fluid, and further deposition stops (Figure 2.3(a) [7]). When deposits have accumulated at the lower superheater region, the heat transfer between the flue gas and the boiler tubes is reduced, leading to an increased flue gas temperature in the higher superheater region. This causes deposits in the higher superheater region to fall within the sticky temperature zone leading to new deposition in this region (Figure 2.3(b) [7]). One can envision a vicious cycle of deposit accumulation as more deposits are shifted to the sticky temperature zone resulting from the continued reduction of heat transfer between the flue gas and the boiler tubes in the superheater region (Figure 2.3(c) [7]). The deposits in this superheater region usually form heavily on the leading edge of tubes, and may eventually bridge the spacing between adjacent superheater platens (Figure 2.4 [7]).
Generating Bank: The deposit accumulation in the higher superheater section eventually leads to plugging at the boiler bank inlet, where the flue gas passage is narrower than the superheater. However, at the center of the boiler bank, fume deposition is the most dominant since the majority of carryover particles have already been screened out upstream and also since the steam temperature is much cooler, favoring condensation buildup on the surface of the boiler tubes. As the deposit layer thickens, its outer surface temperature rises and sintering occurs leaving hard deposits on the outer layer while the inner layer may still remain unsintered and soft. These deposits can be removed by sootblowers due to the weak bond between the inner layer of the deposit and the tube [7].

Economizer: Deposit accumulation leading to plugging can also occur in the economizer but the exact cause of plugging is not clear. It is unlikely due to carryover as the amount of carryover is small and the flue gas temperature is too low to make carryover sticky [7].
2.1.2 Deposit Removal by Sootblowing

Sootblowers can remove deposit accumulation by at least four different mechanisms – (i) breakup due to high internal stresses caused by jet pressure, (ii) removal by debonding from tube surfaces, (iii) removal due to tube bending or vibration, (iv) thermal shock [16]. These mechanisms will be discussed in greater detail in the following section.

Breakup due to high internal stresses caused by jet pressure. Kaliazine et al. [16] performed experimental studies using lab-scale experiments and model deposits made from gypsum, with a supersonic air jet. Their analysis showed that for a wide range of brittle deposits, the breakup depends only on the peak impact pressure (PIP) of the jet and the deposit tensile strength (\(\sigma_t\)). They estimated that the deposit will be removed if the peak impact pressure exceeds a critical value equal to two times the tensile strength.

\[
PIP \geq 2\sigma_t \quad \ldots (2.1)
\]

However, their experimental work with larger deposits indicates that the failure criterion presented above deviates from this and that the failure criterion depends largely on the deposit-tube interface where adhesion strength is more dominant.

Removal by debonding from tube surfaces. In general, a deposit is removed by debonding when the stress generated by the impact of the sootblower jet at the interface of the boiler tube exceeds the adhesion strength of the deposit. The larger the adhesion strength, the more difficult it is to detach a piece of deposit from the tube surface. It is then desirable to estimate both the stress at the interface of the boiler tube from the impact of the sootblower and the adhesion strength of the deposit to predict whether a deposit may be easily removed by debonding. Although the stress
distribution at the interface can be calculated by means of the theory of elasticity, it is not feasible since the shape, exact pressure fluctuations, and mechanical properties of the deposit are not well known [17]. Kaliazine et al. [17] developed a theoretical model for an estimate of the stress distribution at the tube-deposit interface \( \sigma_{max} \), that does not require the exact deposit shape or complete distribution of outer forces over the deposit surface.

\[
\sigma_{max} = \frac{F}{LR} \psi_{max} \left( \alpha_0 \frac{M}{FR} \right) \quad \text{(2.2)}
\]

Where \( \alpha_0 \) is the tube sector angle, \( R \) is the tube radius, \( L \) is the deposit length measured along the tube axis, \( F \) is the net force applied to the deposit in the transverse direction, \( M \) is the net moment of force about the axis of the tube, and \( \psi_{max} \) is a dimensionless function. A laboratory study was also conducted to investigate the adhesion strength of molten precipitator

![Figure 2.5. Adhesion strength versus boiler tube temperatures in different zones of the recovery boiler.](image-url)
dust dripped onto a steel plate heated to various temperatures, which is characteristic of boiler tubes (refer to Figure 2.5) [17].

The adhesion strength ($\sigma_{adh}$) was calculated as:

$$\sigma_{adh} = \frac{4F}{\pi D_f^2} \quad \ldots (2.3)$$

Where $D_f$ is the inside diameter of the steel ring. A relatively simple relationship was then developed to relate PIP and the debonding of deposits.

$$PIP \geq \frac{1}{\psi_{max}} \sigma_{adh} \frac{d}{H} \quad \ldots (2.4)$$

Where $d$ is the tube diameter, and $H$ is the deposit thickness. This relationship shows that the removal by debonding requires a smaller PIP as the deposit thickens.

**Removal due to tube bending or vibration.** In the superheater section of the boiler, tubes are arranged in platens. A platen is a tube sheet with in-line tubes of zero front-to-back spacing. These platens are held on supports and suspended from the boiler ceiling making them free to swing (Figure 2.6 [8]). As the sootblower jet impinges on these platens, the superheater platens can swing. Experimental work has shown that the platens swing with a very low frequency, around 0.2 Hz, but with a high amplitude (25 to 60 N) [18]. This swinging causes the boiler tubes to vibrate.
bend and creates both compressive and tensile stresses on the deposit which can eventually fatigue and crack them.

**Thermal shock.** When fouling is more severe and sootblowing becomes ineffective, a technique called “chill and blow” is employed to clean the boiler. Instead of spraying black liquor, an auxiliary oil is used instead to decrease furnace temperature to 300 °C and eliminate any further deposit accumulation. This sudden drop in flue gas temperature rapidly cools deposits, causing them to contract faster than the steel boiler tubes [19]. This consequently cracks deposits, resulting in its detachment from the boiler tubes. Martinez et al. [20] proposed that the linear expansion coefficient of deposits is approximately three times larger than that of carbon steel, suggesting that deposits do indeed contract more than the boiler tubes.

### 2.2 Fundamental Hydrodynamics of Sootblowing

Sootblowers use converging-diverging nozzles to generate supersonic steam jets, as a way to remove deposits from boiler tubes. They use supersonic nozzles in place of subsonic ones for good reason, justified by comparing the peak impact pressure (PIP) along the jet centerline between a fully expanded supersonic nozzle and a fully expanded subsonic nozzle. Emami et al. [21] simulated both these nozzles having the same mass flow rate and showed clearly that the PIP for a supersonic nozzle is substantially higher than for a subsonic one (Figure 2.7 [21]).
2.2.1 Nozzle Design and Flow Regime for Sootblower Nozzle

A supersonic jet is achieved by designing a converging-diverging nozzle with a shape having a throat diameter that is the narrowest part of the nozzle. For a compressible gas flow, when the ratio between the inlet pressure \( p_o \) and the atmospheric pressure \( p_\infty \) exceeds a critical value, the velocity of the fluid at the throat reaches the local speed of sound for that fluid (Figure 2.8)[14].

**Figure 2.7.** Comparison of supersonic nozzle with equivalent subsonic nozzle.

**Figure 2.8.** Cross section of converging-diverging nozzle.
The critical ratio is:

\[
\left(\frac{P_o}{P_{\infty}}\right) = \left[\frac{(\gamma + 1)}{2}\right]^{\frac{\gamma}{\gamma-1}} \quad \ldots (2.5)
\]

Where \(\gamma\) is the ratio of the specific heats for the gas \((c_p/c_v)\), \(c_p\) is the specific heat at constant pressure, and \(c_v\) is the specific heat at constant volume. Since a sootblower exceeds this critical value, the steam exiting the nozzle will exceed the speed of sound.

There are two basic parameters that control the PIP profile along the axis of the jet: the inlet pressure and the throat diameter [17]. These parameters can be readily controlled and provide flexibility in designing a fully-expanded nozzle. A fully-expanded jet occurs when the nozzle exit pressure, \(p_e\), is allowed to adjust exactly to the ambient pressure, \(p_{\infty}\) (Figure 2.9(a) [8]). When the nozzle exit is not equal to the ambient pressure, the jet is said to be either overexpanded \((p_e < p_{\infty})\) (Figure 2.9(b) [8]) or underexpanded \((p_e > p_{\infty})\) (Figure 2.9(c) [8]). If the jet exit pressure is not equal to the atmospheric pressure, a series of oblique expansion and compression waves occur. These waves reduce the kinetic energy of the jet and in turn the PIP. If the jet is very underexpanded, meaning the exit pressure is much greater than the atmospheric pressure (i.e. \(p_e > 2p_{\infty}\)), the expansion is accompanied by shock waves converting kinetic energy into internal energy and causing a further reduction of kinetic energy of the steam. It is thus desirable to design a jet that is fully expanded to maximize the energy leaving the nozzle [14]. A fully-expanded jet can be
designed with some flexibility and experimental work has shown that increasing the nozzle throat diameter causes the jet to decay more slowly, thereby increasing the penetration depth of the jet, while maintaining PIP near the nozzle exit. This may be desirable in areas of the boiler where deposit buildup is further away from the sootblower. On the other hand, if the inlet pressure to the nozzle is increased, the maximum PIP at the outlet of the nozzle is larger, however, the jet penetration distance is smaller. This may be desirable in order to remove hard and sintered deposits that form on the leading edge of the superheater tubes [17].

The divergent section of the nozzle has also been found to have a profound effect on the PIP delivered at the exit of the nozzle [7]. In the past, sootblowers have been designed to have a short distance between the throat and the nozzle exit, making it impossible for the jet to adjust to its exit pressure. A careful redesign of the divergent section of the sootblower nozzle in the past decade has allowed the jet to reach full expansion. This redesign included a differing contour shape and a larger distance between the throat and nozzle exit (refer to Figure 2.10 [7]).

![Figure 2.10. Development of sootblower nozzle.](image)
2.2.2 Mechanics of Deposit Removal

The cleaning potential of the sootblower has been correlated with its PIP. As mentioned previously, this is the pressure measured by a pitot tube inserted into the jet at its centerline. Kaliazine et al. [17] explored the feasibility of using low-pressure steam for sootblowing to reduce the consumption of valuable high-pressure steam while maintaining comparable cleaning potential. It was found that a fully expanded jet at 80% flowrate has a comparable PIP to a fully underexpanded jet at distances beyond 1.25 m. Similarly, the fully expanded jet at 60% flow rate can provide competitive PIP, but only at distances closer than 0.6 m (refer to Figure 2.11) [17].

![Figure 2.11. Decay of peak impact pressure for both underexpanded nozzles and fully expanded nozzles.](image)
When a sootblower hits a deposit, it produces a longitudinal force in the axial direction of the jet commonly referred to as drag force as well as a lateral force perpendicular to the jet commonly referred to as lift force. The aerodynamics are difficult to predict since they are influenced by the shape of the deposit, velocity of the jet, turbulent pulsation of the jet, and eddy shedding from the tube [17]. These factors make it difficult to predict the interactions between the jet and the deposit. The following sections will discuss the experimental and numerical work done to date at the University of Toronto aimed to characterize the jet and its interaction with deposits inside the recovery boiler.

2.3 Previous Experimental Studies

2.3.1 In-Situ Measurements of Sootblower Jet

One of the first experimental studies aimed to examine the performance of a sootblower inside a recovery boiler was done using an in-situ jet force measurement system (refer to Figure 2.12 [22]). The force measurement system was installed inside newly commissioned cold boilers, so that no deposit buildup had previously occurred. As the jet impinged on the target, the force transducer measured the jet force of the sootblower [18, 22].
Two types of sootblowing tests were performed: stationary and running sootblowing. The stationary sootblowing test locked the sootblower from rotating while the running sootblowing test mimicked more realistic operating conditions as it allowed the sootblower to rotate.

![Figure 2.12. Force measurement system.](image)

![Figure 2.13. Jet force as a function of lance pressure.](image)
Figure 2.13 [22] shows the average force obtained at the Obbola Mill as a function of lance pressure with varying probe diameters and distances away from the nozzle exit.

2.3.2 Scaled-down Experiments on Brittle Deposits

Since the recovery boiler operates at high temperatures and poses a hostile environment for sensor equipment, the majority of the experimental work related to sootblowing performance has been done using a ¼ scale laboratory apparatus of the nozzle and boiler tube arrangements to maintain a reasonable size and complexity. Pophali et al. [23] visualized the breakup mechanisms of model brittle deposits using high-speed photography (refer to Figure 2.14 [23]).

![Figure 2.14. Breakup of model brittle deposits.](image)

The experimental work identified three separate mechanisms by which brittle deposits break, based on a critical diameter ratio between the jet and the deposit, as described in Table 2.1 [8]. The work found that with thin deposits, cracks form easily and breakup occurs rapidly, while for thick deposits cracks do not readily form.
2.3.3 Flow Visualization of Sootblower Jet

Using a similar apparatus as the one presented above, Pophali et al. [8], used a schlieren technique coupled with high speed photography to visualize the interaction of a supersonic jet with models of tube arrangements found in the superheater, generating bank, and economizer sections of the recovery boiler. The schlieren technique, invented by German physicist August Toepler, is based on the principle that parallel light rays refract when they pass through inhomogeneities such as the density gradients within a supersonic jet (refer to Figure 2.15 [8]). A set of concave mirrors and light sources were arranged to take advantage of this principle and visualized supersonic jets interacting with different tube bundles. The most significant contribution of this experimental work

<table>
<thead>
<tr>
<th>Observed Breakup Mechanism</th>
<th>Jet-to-deposit Diameter Ratio</th>
<th>Breakup Image</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial crack formation</td>
<td>$d/d_{\text{deposit}} &gt; 0.51$</td>
<td></td>
</tr>
<tr>
<td>Surface erosion + axial crack formation</td>
<td>$0.36 &lt; d/d_{\text{deposit}} &lt; 0.51$</td>
<td></td>
</tr>
<tr>
<td>Surface erosion + spalling</td>
<td>$d/d_{\text{deposit}} \leq 0.36$</td>
<td></td>
</tr>
</tbody>
</table>

Table 2.1. Breakup mechanisms of brittle deposits.
was that it characterized both the primary and secondary jets produced by the sootblower interacting with boiler tubes. As the jet impinged on a cylinder, a weaker supersonic jet, known as the secondary jet, deflected at an angle that depends on the eccentricity of the primary jet with the cylinder (refer to Figure 2.16)[8].

![Figure 2.16. Visualization of supersonic jet with superheater platens.](image)

### 2.4 Numerical Models of Sootblower Jet

More recently, with the development of numerical modeling software, computational fluid dynamics (CFD) has been used to model the flow of sootblower jets interacting with tube banks and deposits over varying conditions encountered during sootblowing. In principle, the numerical model can describe the flow of the jet by solving only the Navier-Stokes equation; however, this would require an impossibly fine grid and a very small time step to accurately keep track of the detailed turbulent motion of the jet. As a way to overcome this problem, turbulence modeling is used instead which takes an average of the Navier-Stokes equations, introducing several additional terms. Although the most common turbulence model used is the $k-\epsilon$ model, its predictive ability
deteriorates for supersonic flow applications since it was originally designed for incompressible flows [24].

2.4.1 Modeling Sootblower Jets and Impact On Deposit Removal

The majority of the CFD models developed to simulate sootblower jets are based on the CFDLib code obtained from the Los Alamos National Laboratory and initially focused on modeling free supersonic jets with no obstacle interactions [12]. They modeled both fully expanded and more complicated underexpanded supersonic jets by applying corrections to the k-\( \epsilon \) model for better predictions.

Having modeled free jets, additional studies by Emami et al. [12] and Doroudi et al. [13] used CFD to simulate the interaction between a sootblower jet and the convective sections of the recovery boiler (refer to Figure 2.17 [12] and Figure 2.18 [13]).

![Figure 2.17. Comparison of Schlieren images and numerical results of sootblower jet interacting with lab-scale generating bank geometry.](image1)

![Figure 2.18. Comparison of Schlieren images and numerical results of sootblower jet interacting with lab-scale economizer geometry.](image2)

More recently, the pressures predicted by CFD on model deposits were exported and used as boundary conditions to both FEA (refer to Figure 2.19 [11]) and FDEM (refer to Figure 2.20 [11])
solvers in order to simulate the stress distribution within a deposit [11]. The FEA study by You et al. [11] shows that a compressive field occurs at the impinging surface of the deposit, while a tensile field occurs at the deposit-tube interface.

![Figure 2.19. Static FEA model.](image)

![Figure 2.20. FDEM model at time 104 µs.](image)

This suggests that a crack may form at the inner surface of the deposit and work its way to the outer surface. This was similarly predicted by the FDEM software as well.

### 2.4.2 Validation of the Numerical Models

As a way to validate the numerical work of free jets, the model results were compared to pitot tube measurements of the peak impact pressures obtained by Norum and Siener [25] (refer to Figure 2.21 [26]), as well as the force measurements obtained from the probe inside of newly commissioned boilers (refer to Figure 2.22 [26]), discussed in Section 2.3.1. It was found that the original CFDLib code did not yield accurate predictions of simple fully-expanded jets and required modifications to the k-ε models [12]. Tandra et al. [27] modified the turbulence model by adding compressibility corrections that yielded better agreement with select measurements of fully expanded free jets and jet flows between platens, referred to as the SJT model. However, although the SJT model predicted better agreement with fully expanded free jet data, it failed to accurately
predict the complex nature of underexpanded jets, which involve multi-cell shock waves. This led to the development of the SJT-shock model which made further corrections by imposing a realizability condition [28] and taking into account shock unsteadiness effects [29].

![Normalized pressure along centerline axis of supersonic jet.](image1)

**Figure 2.21.** Normalized pressure along centerline axis of supersonic jet.  

![Results obtained from in-situ measurement device.](image2)

**Figure 2.22.** Results obtained from in-situ measurement device.

Although experimental studies were used to validate and improve the free jet models, there has been little experimental work to validate predictions of the sootblower interaction with boiler tubes and deposits. The validation studies to date have only focused on qualitatively comparing

![Comparing the gross features of the Schlieren images with the numerical models.](image3)

**Figure 2.23.** Comparing the gross features of the Schlieren images with the numerical models.
the gross physics of the shock and expansion waves in both the primary and secondary jets with the Schlieren images obtained by Pophali et al. [26] (refer to Figure 2.23 [12]). Since the FEA and FDEM analysis also depends on the accuracy of the predicted pressures from CFD, it is unclear if the computed internal stresses by FEA and FDEM analysis are accurate.

2.5 Conclusions

The ability of the CFD to accurately predict the fluid mechanics of the sootblower jet is of great importance not only from the perspective of analyzing different sootblower conditions, but also since its accuracy has a direct impact on the ability of the FEA and FDEM analyses to correctly predict deposit failure. Although sootblowers are designed to be fully expanded, it is sometimes difficult to achieve such a state in practice. As a result, accurate simulations of underexpanded jets are required to closely mimic recovery boiler conditions. With the complex nature of underexpanded jets, CFD predictions become increasingly more difficult. Experimental studies have been conducted in the past that visualized the breakup process of model brittle deposits as well as the flow of a supersonic jet, which were used to validate these CFD models. However, they are limited by their qualitative nature, and provide only visual validation of the gross features of CFD. Furthermore, the quantitative analysis experimentally performed to date is incomplete, since the comparisons focus primarily on free jet models.

With the added complexity of the interaction between sootblowers and deposits, further experimentation is needed to validate these CFD models. The following section will discuss the experimental design used to validate the CFD predications of complex underexpanded jets as they interact with model deposits.
CHAPTER 3

EXPERIMENTAL DESIGN AND METHODOLOGY

The experimental design described in this section will provide a quantitative measure of the reaction forces of a jet as it interacts with different model deposits, as a way to provide validation of the current CFD models. The experimental setup uses a scaled-down version of an actual sootblower, with a supersonic nozzle representing the sootblower (refer to Figure 3.1), and will be discussed in more detail in the following chapter.
3.1 Physical Apparatus

The experimental setup shown in Figure 3.2 was custom designed and used to measure the reaction forces as a jet interacts with tubes and deposits. Similar flow characteristics were modeled in CFD and the output was compared to the experimental results as a way to validate the models. The apparatus consists of a fixed nozzle mounted on a manual x-y-z stage and two track roller carriages with a guide rail assembly, each secured to one of the elevated horizontal beams.

![ISOMETRIC VIEW:](image1.png)  
![CROSS SECTIONAL:](image2.png)

**Figure 3.1.** Machined converging-diverging nozzle representing sootblower.

**Table 3.1.** Nozzle and flow details.

<table>
<thead>
<tr>
<th>Nozzle Details</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Throat Diameter ($d_t$)</td>
<td>4.5 mm</td>
</tr>
<tr>
<td>Exit Diameter ($d_e$)</td>
<td>7.4 mm</td>
</tr>
<tr>
<td>Exit Mach Number ($Ma_e$)</td>
<td>2.53</td>
</tr>
<tr>
<td>Exit Density ($\rho_e$)</td>
<td>2.72 kg/m$^3$</td>
</tr>
<tr>
<td>Nozzle Inlet Pressure ($p_o$)</td>
<td>1,893 kPa gauge (260 psi)</td>
</tr>
<tr>
<td>Flow Rate</td>
<td>59.4 g/s</td>
</tr>
</tbody>
</table>
The track roller carriages are nearly frictionless and translate freely on the guide rails. Pillow block bearings are attached to the top of each track roller carriage, and a rotary shaft that represents a boiler tube is mounted in the pillow block bearings. On one end of the rotary shaft there is a torsion sensor (Futek Model TFF400), and a normal sensor (Futek Model LSB200) is attached to the end of each elevated guide rail assembly.

Compressed air flows through the converging-diverging nozzle, creating a supersonic air jet that impinges on the

**Figure 3.2.** Experimental design.

**Figure 3.3.** Close-up of experimental design.
rotary shaft. This causes the track roller carriage assembly to slide and press against two normal sensors located at the ends of the elevated guide rails. These normal sensors measure the horizontal reaction force (drag force), and the torsion sensor measures any moment (lift force) produced by the supersonic air jet (refer to Figure 3.3). The position of the nozzle is controlled by the x-y-z stage in the both the axial horizontal and vertical directions of the jet. For more precise positions in the vertical direction, an electronic digital reader was attached to the x-y-z stage.

The compressed air was supplied by a Makita compressor (Model AC310HX) into a Manchester air receiver (76 L) that was sized to supply an inlet pressure of 260 psig to the nozzle for a 2s time period. A solenoid valve (manufactured by ASCO) was triggered to control the release of air, during which time the normal and torsion sensors recorded the reaction forces. The byproduct, however, of compressing air at such a high pressure is that it condenses during its cooling process (refer to Appendix A for a moisture calculation). This moisture can damage instrumentation, corrode the air receiver, and is not included in the CFD modelling. As a result, a panel was designed (manufactured by Swagelok) to remove this moisture by means of a desiccant air dryer (refer to Figure 3.4). The effectiveness of the desiccant is indicated by a color change, from blue to pink, and can be seen by a built-in sight glass as shown in Figure 3.4. When the bulk of the desiccant reached saturation, it was removed and heated in an oven at 255 ºF for 12 hours in order to be reactivated [30]. Since the desiccant gels corrode over time, an air filter was added downstream of the air dryer.
**Typical Operational Steps:** A combination of shut-off valves allowed ease in controlling the air flow in the desired direction depending on the time of experimentation (refer to Figure 3.4). When the second and third shut-off valves were closed and the first shut-off valve was opened, the Manchester air receiver was filled by the compressor. Once the air receiver reached the setpoint pressure, the first shut-off valve was then closed, and the second shut-off valve was opened, allowing compressed air to flow from the Manchester air receiver to the solenoid valve. The nozzle inlet pressure was maintained at 260 psig for all experimental runs. In each experiment, the solenoid valve was activated for 2s allowing air to impinge on the rotary shaft for this time period. This duration was found to be sufficient considering the time required for the jet to reach steady state and the amount of air used per experiment. At the completion of all experimental runs, the
air was depleted to the environment from the air receiver and all piping, by means of the third shut-off valve.

### 3.1.1 Scaled-down Nozzle and Deposit Geometry

The main components of the experimental apparatus were ¼ scale models. This scaled down factor has been used in related experimental work and as a result was similarly used here to reduce size and complexity [16, 23].

The model deposits were similarly scaled, and 3D printed in a hard laser-cured polyurethane using a Form 2 3D printer, manufactured by FormLabs. Two types of deposits were designed and printed: symmetrical and asymmetrical deposits; and their respective design details can be seen in Figure 3.5. The modeling software used to design these deposits was Autodesk Fusion 360, as it has direct integration with the PreForm software which allows the final design model to be oriented, supported, and laid out before being 3D printed. The limitation of the Form 2 build volume meant that the deposit could not be printed as one body but rather as three separate components, each 3 inches in length, that were connected with superglue once printed. During various printing trials, it was found that the best orientation for maintaining print quality and alignment was to orient the deposits vertically and have them printed directly on the build platform without any initial supporting layers. Since there was no way of tightly securing these deposits to the rotary shaft without drastically obstructing the flow of the jet, #2-56 screws were used on the backside of the

![Figure 3.5. Deposit geometries.](image-url)
deposit to clamp deposits tightly onto the rotary shaft. This was achieved by printing threads as well as designing a star-shaped cavity within the deposit for reduced rigidity and increased flexibility. That way, when the screws were tightened, the deposit was flexible enough to bend without cracking. Figure 3.6 shows a typical 3D printed deposit that is clamped onto the rotary shaft by the screws.

### 3.1.2 Similarity of Lab Air Jet to Actual Sootblower Jet

It was important that the experiment be both geometrically similar to actual sootblower conditions, and also dynamically similar. Sootblowers use steam instead of air which could not be used in this study for reasons of safety and feasibility. However, both superheated steam and air are homogeneous fluids that have similar specific heat ratios, making air an ideal fluid to use in a laboratory environment. For the laboratory jet to be dynamically similar to the sootblower, it needs to have a comparable exit Mach number and Reynolds number. The Mach number is a function of the ratio of the nozzle exit diameter to the throat diameter, and since the converging-diverging nozzle was a $\frac{1}{4}$ scale model of an actual nozzle, the Mach number of the jet was similar to that of a sootblower jet. Using a one-dimensional gas dynamics relationship, the exit Mach number of the
air jet was computed to be 2.53, which is similar to that of an actual sootblower (refer to Appendix B for an in-depth calculation). Pophali et al. [23] found that the Reynolds number for the lab experiments and the actual sootblower were also similar. The Reynolds number for the lab experiments was $1.26 \times 10^6$, whereas for an actual sootblower it is $1.81 \times 10^6$. A summary of the similarity between the supersonic nozzle and the actual industrial sootblower is shown in Table 3.2.

<table>
<thead>
<tr>
<th></th>
<th>Sootblower</th>
<th>Experimental Nozzle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Pressure ($p_o$)</td>
<td>200 – 260 psi</td>
<td>260 psi</td>
</tr>
<tr>
<td>Exit Mach Number ($Ma_e$)</td>
<td>2.2 – 2.8</td>
<td>2.5</td>
</tr>
<tr>
<td>Reynolds Number</td>
<td>$1.81 \times 10^6$</td>
<td>$1.26 \times 10^6$</td>
</tr>
<tr>
<td>Specific Heat Ratios ($k = C_p/C_v$)</td>
<td>1.3</td>
<td>1.4</td>
</tr>
</tbody>
</table>

### 3.2 LabVIEW Control and Data Acquisition (DAQ) System

A control and data acquisition system was used to control the two normal sensors and the torsion sensor, as well as operate the solenoid valve. The control system consisted of a laptop running Windows 10 with National Instruments (NI) LabVIEW 2015 software installed, and was used to control the NI 6001 multifunction data acquisition module that generated an analog voltage by the normal and torsion sensors. A schematic of the data acquisition control system is presented in Figure 3.7. The graphical program coded in LabVIEW communicated with the data acquisition to allow the solenoid valve to be opened at the same time that all of the sensors started recording the analog voltage. The air was allowed to flow through the solenoid valve and impinge on the rotary shaft or deposit for 2s. After this time period, the solenoid valve was closed, and the sensors stopped recording any more data. At the end of each experimental run, the air receiver decreased
to roughly 250 psig; it was then filled once more to 260 psig so that all runs had the same supply pressure.

3.3 Sensor Calibration

Since the sensors record only an analog voltage, they must be calibrated to output a normal force and moment instead. As a way to calibrate the sensors, a simple pulley system was designed to apply known loads to the sensors as can be seen in Figure 3.8. A string was wrapped numerous times at each end in order to be taut; one end was wrapped around the rotary shaft and the other end was wrapped around a carabiner. The string rested inside of the pulley assembly and the

![Figure 3.7. LabVIEW control.](image)
carabiner was used as an attachment point for a place holder. A 1000 g weight was added to the place holder, generating an analog voltage reading by the normal sensors and torsional sensor.

Using Newton’s second law of motion and assuming a weightless string as well as a frictionless pulley (refer to Appendix C for a detailed calculation), a calibration ratio was obtained from the calculated forces and the voltage generated by the sensors. For increased calibration accuracy, this calibration method was repeated at each end of the rotary shaft so that each normal sensor was calibrated independently of the other. This generated slightly different analog voltages by the torsional sensor at each end which were averaged and used as the torsion sensors respective calibration ratio.

With the sensors calibrated, they would be used to experimentally output a normal force and moment. Figure 3.9 shows a typical force recording by one of the normal sensors. It can be seen that the sensors experience a large instability in the first few hundred milliseconds when the supersonic jet initially impinges on the rotary shaft or deposit, as well as a shallow decay in force over time. The large instability at the beginning is caused by the transition of the jet from its transient state to its quasi-steady state [8], and the decay of the force is due to the reduction of pressure in the Manchester air receiver over time. Since the CFD simulations were modeled at
steady state, the first few hundred milliseconds and last few hundred milliseconds were not included, and only an average of 79 force readings were taken from 0.52s to 1.3s.

Figure 3.9. Typical sensor recording.

This way, the transient stages of the jet were not considered, and the force can be assumed to be relatively constant in that time period. This was done for all experimental runs so that the presented normal forces and moments are steady state solutions that are not time dependent.

Before any experimental results were obtained, the centerline axis of the jet was aligned vertically relative to the center of the rotary shaft. The symmetrical deposit was placed onto the rotary shaft, and using the torsion sensor, the jet was vertically offset above and below the center of the rotary shaft at a horizontal distance of 4 mm from the surface of the deposit. The vertical offset at which the torsion sensor recorded a zero moment was considered as the center position of the jet relative to its centerline axis.
3.4 Repeatability and Accuracy of Measurements

The uncertainty in the experimental work can be quantified by calculating the standard error. The sensors have a non-repeatability of $\pm 0.001 \, mV/V$ as specified by the manufacturer. As a measure of the uncertainty of the sensors, the experiment was repeated seven times for each experimental run as the jet impinged on the rotary shaft and was moved further away in the axial direction relative to the centerline axis of the jet. The standard error was then calculated and provided insight that the sensors were very repeatable. On the basis that the sensor uncertainty was minimal, and the results were repeatable, all experimental runs performed thereafter were repeated only twice and the standard error bars were not plotted since they were insignificantly small.

3.5 Numerical Model

The reaction forces measured by the custom designed experimental apparatus were compared to the results of a CFD model that had a similar geometry and flow characteristics, using the ANSYS CFD V18.2 Fluent software package. The simulations were carried out in a 3D axisymmetric domain using an ideal gas model for air and pressure-based boundary conditions. For the walls, a no slip condition was imposed to allow for wall friction.
The computational domain was $20 \times 24 \times 10 \, D$, where $D$ is the nozzle exit diameter. Figure 3.10 shows a schematic of the computational domain and boundary conditions. As seen in the Figure, a symmetry boundary was applied along the jet centerline, and far from the jet an ambient pressure was specified. A structured mesh was created using ANSYS ICEM CFD, with a finer mesh close to the jet centerline that became progressively coarser further from the jet core. Figure 3.11 shows a typical structured mesh for the free jet case. The mesh size for the simulation results presented in this thesis was approximately 2,000,000 cells, and the run time was around 10 hours to reach convergence.

**Figure 3.10.** Computational domain and boundary conditions.

**Figure 3.11.** Typical mesh design.
As a way to validate the simulation setup and cell size, a free jet simulation using the mesh shown in Figure 3.11 was run in an underexpanded state, and was compared to literature results. Pophali et al. [23] used a similar nozzle geometry; having the same throat and exit diameter but different convergent and divergent section profiles. Since the exact nozzle geometry details were not published, the nozzle geometry designed for this study was used in this comparison. In his experimental work, Pophali et al. [23] obtained peak impact pressure measurements of a free underexpanded jet along its centerline axis. The experimental peak impact pressures obtained by Pophali et al. [23] were compared to the numerical simulation. As seen in Figure 3.12, there is reasonably good agreement between the experimental and numerical results. The differences in the peak impact pressure profiles can be reasonably attributed to the dissimilar convergent and divergent sections of the compared nozzles.

**Figure 3.12.** Comparing experimental versus numerical peak impact pressures: PIP values are normalized by the supply pressure $p_o = 2.14 \, MPa$; spatial coordinates are normalized by the nozzle exit diameter $d_e = 7.4 \, mm$. 
CHAPTER 4

FORCE MEASUREMENTS OF SOOTBLOWER JETS INTERACTING WITH DEPOSITS

When a jet hits the deposit, it produces a longitudinal force, known as the drag force, and an axial force perpendicular to the jet, known as the lift force as seen in Figure 4.1 [17]. The experimental apparatus directly measures the drag force but indirectly measures the lift force since the assembly is not free to move in the direction perpendicular to the jet due to the fixed railing cart assembly. Instead, the torsion sensor measures the moment instead, which is the product of the net force and the moment arm.
The drag force ($F_D$) is comprised of a pressure drag and a viscous drag. As the jet hits the deposit and separates, a pressure drag results due to the pressure difference between the tightly packed air molecules at the front of the deposit and the more loosely packed air molecules at the rear of the deposit [31]. Along with this pressure drag, a viscous drag force also results, due to the friction at the surface of the deposit. These viscous forces create boundary layers at the surface and lead to the eventual flow separation of the jet [32]. When the flow separates from the deposit-tube assembly, circulating regions occur, and a wake is formed behind the deposit.

The lift force ($F_L$), on the other hand, results from an imbalance of pressure caused by a difference in the tangential velocity between the opposing sides of the deposit. Many factors influence the formation of lift forces which include irregularly shaped deposits, and turbulent pulsations of the jet. The net lift force acts in a direction towards where the highest tangential velocity exists, since this is the region with the lowest pressure.

Both lift and drag forces have a direct impact on how deposits are removed by sootblowers. While the pressure drag force pushes the deposit against the tube surface, which may initiate cracks at the interface of the deposit, the viscous drag can create shear stresses at the interface between the deposit and the boiler tube [11]. On the other hand, lift forces might be more effective in

**Figure 4.1.** Sootblower jet interacting with deposit.
removing deposits by debonding [33]. If an imperfection within the deposit such as a crack exists at the interface between the deposit and the boiler tube, the adhesion strength at this interface would be weakened. As such, the stresses generated by the jet may produce a moment that is larger than the adhesion strength at this interface and result in the removal of the deposit by debonding.

The complicated aerodynamics associated with the interactions of the sootblower jet and a deposit will be measured, and the results will be compared to CFD simulations of similar interactions.

4.1 Interaction Between A Jet and Boiler Tube

In the first validation, the reaction forces of an underexpanded jet interacting with a single boiler tube were measured. Compressed air flowed through a converging-diverging nozzle with an inlet pressure of 260 psi creating an underexpanded supersonic jet that impinged on the rotary shaft (boiler tube). The drag force was calculated by a simple force balance; using the measured normal forces at both ends of the rotary shaft.

4.1.1 Jet Forces Exerted on Boiler Tube

Figure 4.2 shows the results of the measured drag force as the jet interacted with the rotary shaft.
The distance from the nozzle exit to the surface of the rotary shaft was varied from 4 mm to 110 mm, which was the limit of the x-y-z stage. As seen in Figure 4.2, there are small force fluctuations present in the first 30 mm from the surface of the rotary shaft, followed by a steady decrease of drag force thereafter.

Both these force fluctuations and decrease in drag force can be explained by referring to the flow phenomena of a free supersonic jet. Since the jet is underexpanded, a series of shock cells comprised of expansion and compression waves occur near the exit of the nozzle, as the jet adjusts to atmospheric pressure, as seen in Figure 4.3. These multi-shock cells cause the observed force fluctuations, while the
turbulent mixing of the jet eventually encapsulates the flow field downstream of the nozzle, explaining the reduction of drag force [21].

Since the rotary shaft is a symmetrical body with a smooth surface, there were no moment recorded by the torsion sensor even when the jet impinged at offsets perpendicular to the centerline axis of the jet.

4.1.2 Comparing Experimental Results with Numerical Models

As a way to validate the numerical model, only one of the above experimental cases was simulated with CFD, since modeling each one requires significant computational time. The experimental case selected was 50 mm from the surface of the rotary shaft with respect to the centerline axis of the jet. The final mesh along with the resulting velocity contour are shown in Figure 4.4 and Figure 4.5, respectively.

Figure 4.4. Final mesh design of jet interacting with boiler tube.

Figure 4.5. Velocity contour of jet interacting with boiler tube.
The drag force predicted by CFD was 16.5 N while the experimentally measured drag force was 16.1 N. This comparison demonstrates the ability of the CFD software to accurately predict the complex nature of an underexpanded jet interacting with a single boiler tube.

4.2 Interaction Between A Jet and Symmetrical Deposit

Since deposits inside of a recovery boiler do not have a specific shape and size, there is no direct reference for their geometry. The first set of experiments involving a jet interacting with a deposit were measured using a symmetrical deposit that was 3D printed. A similar experimental setup and procedure as the one described in Section 4.1.1 was followed, where the distance of the nozzle exit was increased relative to the surface of the deposit with respect to the centerline axis of the jet. Since the deposit was symmetrical, there were no moment recorded by the torsion sensor even when the jet impinged at offsets perpendicular to the centerline axis of the jet.

4.2.1 Jet Forces Exerted on Symmetrical Deposits

Figure 4.6 shows the measured drag force as the nozzle exit was moved further away from the surface of the deposit. As seen in the Figure 4.6, these results show a similar trend to the rotary shaft impingement described in Section 4.1.1 with the main difference being the magnitude of the drag force. The measured drag force was greater when the jet impinged on the symmetrical deposit than when it impinged on the rotary shaft. This is expected since the drag force is both a function of the jet pressure and its impinging area. Although the pressure of the impinging supersonic jet stayed the same, the impinging area for the symmetrical deposit is much larger than that of the rotary shaft and explains the increase in the drag force.
Having obtained measurements of the drag force impinging on a symmetrical deposit, the 50 mm case was simulated using CFD as a way to validate the numerical model. Since this a similar case to the one selected for the boiler tube validation, the mesh design will not be presented here. Figure 4.7 shows the velocity contour simulated by CFD as the jet interacted with the symmetrical deposit. The drag force predicted by CFD was 19.0 N while the experimentally measured drag force was 23.7 N. In comparing these two results, there is a 20% error between the measured and computed drag force which does not agree as well as the previous comparison made with a single boiler tube impingement. The main difference between the two cases is that the rotary shaft is a smooth surface while the 3D printed deposit can be considered a rough one.
4.2.3 Effect of Surface Roughness on Jet Forces

The effect of surface roughness on drag force was investigated both experimentally and using CFD simulations at a distance of 50 mm from the surface of the deposit with respect to the centerline axis of the jet. As a way to mimic a smooth deposit experimentally, electrical tape was wrapped around its surface as seen in Figure 4.8. With this smooth deposit, the measured drag force significantly dropped from the originally measured 23.7 N to 20.7 N; accounting for a 16% reduction and agreed well with the CFD prediction of 19.1 N.

As a way to investigate the effects of surface roughness in the numerical model, surface roughness at the wall boundary of the deposit was incorporated into the CFD simulation. The
boundary condition for wall roughness, however, requires a sand-grain roughness input rather than a conventional surface roughness. A sand-grain roughness value does not have any direct correlation to surface roughness since the profilometers used to measure surface roughness oversimplify the topography of a rough surface, which leads to significant error in fluid flow calculations [34]. With the complexity of numerically computing a sand-grain roughness value for the 3D printed deposit, a trial and error method was employed to demonstrate the effects of surface roughness on the simulated result. It was found that a sand-grain roughness of 5.5 mm was needed to increase the CFD drag force to 23.9 N, in order to match the originally measured experimental drag force of 23.7 N. Although it is difficult to justify such a large sand-grain roughness value, this exercise demonstrates the effect of surface roughness on the numerically predicted drag force. Furthermore, when plotting the velocity contour for this case (as seen in Figure 4.9), it is clear that by increasing the surface roughness of the deposit, the viscous drag forces increase proportionally which in turn causes the flow of the jet to separate sooner. As a result, a larger wake forms at the rear of the deposit and increases the pressure drag further.

![Simulating rough deposit.](image)

**Figure 4.9.** Simulating rough deposit.
In summary, when comparing the experimental result of a smooth deposit to the originally simulated result, the error was 8% and is within acceptable range. Another contributor to the error in comparing the numerically predicted drag force is not modeling the screws used to clamp the deposit. A simulation incorporating the screws in the numerical model to analyze their effects, and the results of this work, can be found in Appendix D. Since the screws introduce an insignificant amount of error, they were not considered further in any other numerical comparison.

4.3 Interaction Between A Jet and Asymmetrical Deposit

As a way to mimic more realistic operating conditions inside of a recovery boiler, deposit asymmetry and rotation of the sootblower were considered. Since deposits buildup on the leading edge of the superheater platen, an asymmetrical deposit was 3D printed. On the other hand, since the nozzle in the experimental design is fixed in the vice assembly and cannot rotate, a trigonometric relationship was developed to relate the rotating lance position to the fixed nozzle position.

Figure 4.10. Defining rotation radius of lance.
As the lance traverses into and out of the boiler, it rotates about its axis, which has a radius of $R_L$ (refer to Figure 4.10). If the lance rotates at an angle $\varnothing$, at a distance of $D_p$, from the center line axis of both the rotary shaft and the lance, then the nozzle position relative to the asymmetrical deposit will need to be rotated similarly in the experimental design, as seen in Figure 4.11.

Since this is not possible as the nozzle is fixed in the x-y-z stage, the following trigonometric relation was developed to relate the rotation of the lance with the fixed position of the nozzle in the experimental design. As seen in Figure 4.12, if the lance rotates at an angle $\varnothing$, then the asymmetric deposit will have to be rotated at an equivalent angle, as well as translated and offset in the $x$ and $z$ directions respectively, based on the following relationship.
\[ x = D_p \cos(\theta) - R_L \quad \text{(4.1)} \]

\[ z = D_p \sin(\theta) \quad \text{(4.2)} \]

Where \( x \) is the distance from the nozzle exit to the center line axis of the rotary shaft; \( z \) is the distance from both the centerline axis of the jet and the rotary shaft.

This concept of rotating the asymmetrical deposit and translating and offsetting it both in the \( x \) and \( z \) directions to mimic the rotating lance was used in strategically planning the experimental work. The rotating angles considered were \( 0^\circ, 22.5^\circ, 45^\circ, 90^\circ \), and Figure 4.13 indicates the perpendicular offsets selected. The perpendicular offsets with respect to the centerline axis of the jet were determined such that they were equidistant from one another in either the positive or negative direction from the center of the rotary shaft. At each one of these perpendicular offsets, the distance from the nozzle exit to the surface of the deposit was increased from 4 mm to...
110 mm with respect to the centerline axis of the jet, as done in the previous experimental work presented above.

4.3.1 Jet Forces Exerted on Asymmetrical Deposit

The following section will present only the findings for a non-rotated deposit (0°) and a deposit that has been rotated 45°; the remaining data is presented in Appendix E. These two rotation angles were selected since for a non-rotated deposit, since the symmetry about the x-axis provides

Figure 4.13. Measured rotation and perpendicular offsets.
validation of the torsion sensor’s accuracy, while at a 45º rotation, the largest drag and moment forces were recorded by the normal and torsion sensors.

The drag forces measured by the normal sensors for a non-rotating deposit are shown in Figure 4.14. A similar trend as seen in the previous experimental work was observed between

![Figure 4.14. Drag force of jet interacting with an asymmetrical deposit at a non-rotated deposit position.](image)

(2.38) mm and 2.38 mm offsets. In each case, force fluctuations occur near the nozzle exit, followed by a steady decay of drag force as the nozzle is moved further from the surface of the deposit with respect to the centerline axis of the jet. However, when the jet impinged at an offset of 7.13 mm from the center of the rotary shaft, the measured drag force was less than the ones
measured between (2.38) mm and 2.38 mm offsets, especially at distances closer than 22 mm from the surface of the deposit. As the nozzle was moved beyond 22 mm from the surface, the drag force increased proportionally and recorded the largest measured force, before plateauing at roughly 58 mm away from the surface. Thereafter, the drag force decreased until it reached the outmost limit of the vice assembly where it had a comparable force to the other offsets. This increase in drag force between 22 mm and 58 mm at an offset of 7.13 mm can be attributed to the Coanda effect, illustrated in Figure 4.15 [8]. The Coanda effect is a phenomenon that occurs when the jet is mostly tangential to a curved surface [8]. Since the turbulent nature of the jet entrains the flow surrounding it, a curved surface close to the jet causes a low-pressure region between the curved surface and the jet. As a result, the jet adheres to the curved surface, causing a further reduction of pressure on the back side of the deposit. As a result, a larger pressure differences exists between the front and rear of the deposit, which increases the drag force relative to the centerline axis of the jet. At distances closer than 22 mm, the turbulent nature of the jet is far too high for the jet to adhere to the surface but rather the supersonic jet deflects as it impinges on the deposit, which explains the reduced drag force at those distances. Finally, when the nozzle is moved further away than 58 mm, the effect of turbulence is weakened and is not sufficient to create this low-pressure region near the surface.

Figure 4.15. The Coanda effect.
is important to note that when the jet impinged at an offset height of (6.81) mm from the center of the rotary shaft, the measured drag force remained steady as the nozzle was moved further away from the surface of the deposit. This result suggests that the jet is impinging above the expected offset of (6.81) mm; this discrepancy will be explained later in this thesis.

The moments measured by the torsion sensor for the same non-rotated deposit are shown in Figure 4.16. The moment is the product of the net force the deposit experiences and the moment arm, which is defined as the perpendicular distance from the fixed pivot (center of the rotary shaft) to the line of action of this net force. The torsion sensor was calibrated to record a positive moment in a counter-clockwise direction relative to the direction of the jet, as shown in Figure 4.16. It can be observed that the moments generated by the torsion sensor mirror each other about the x-axis (Figure 4.12). This result is expected since for a non-rotated deposit, the deposit is symmetrical.
about the x-axis. When the jet impinges directly at the center of the rotary shaft, the velocity distribution above and below the deposit are similar, and therefore the jet exerts the same pressure on the opposing sides of the deposit, thus generating a net zero pressure with no moment or lift forces recorded by the torsion sensor. However, when the jet impinges at an offset from the center, the magnitude of the moment measured increases in proportion to the offset distance. This is both a function of the net force, which is the pressure exerted over the surface area of the deposit, and the length of the moment arm from the fixed pivot. As seen in Figure 4.17, in both of these scenarios, the line of action passes below the pivot point which forces the moment to act in a counter-clockwise direction, thus recording a positive moment. Therefore, one can infer that when the line of action is below the deposit, the direction of the moment will act in a positive direction, and that the opposite is true when the line of action is above the pivot. As the deposit is moved further away from the nozzle exit, the magnitude of the moment decreases since the turbulent nature of the supersonic jet reduces its velocity, and similarly the net force acting on the deposit.

Figure 4.17. Magnitude and direction of moment.
Figure 4.18 and Figure 4.19 shows the drag force and moments measured by the normal and torsion sensors for a 45° rotated deposit. These offsets were chosen since they represent the extreme cases measured by the sensors. It can be observed in Figure 4.19 that the Coanda effect influences the drag force at an offset of 15 mm, which is the distance most tangential to the curved surface; this is similar to the results presented for a non-rotated deposit. The largest drag force that the normal sensors experienced was at a 7.5 mm offset from the center of the rotary shaft, which happens to be a central offset from the datum (center of rotary shaft) and the most tangential offset height. This is expected since at this offset, the jet encounters the largest impinging area. As a result, the pressure difference between the front and the rear of the deposit is the greatest, causing a large pressure drag force. Figure 4.19 shows the moment forces measured by the torsion sensor at a 45° rotation. Since the pressure drag is the largest at a 7.5 mm offset, it is understandable that the magnitude of the moment will also be greatest at this offset. The direction of the measured
moment makes sense since the line of action is below the pivot, forcing a counter-clockwise direction and a positive recorded moment.

In comparing the magnitudes of both the lift and drag forces, it is clear that the moments and hence the lift forces are much smaller than the drag forces. These results suggest that the removal of these deposits will be dominated by a pressure drag; the effect of pushing the deposit against the tube surface rather than by debonding which is influenced by the lift forces.

### 4.3.2 Comparing Experimental Results with Numerical Models

As a way to validate the complex nature of the underexpanded jet interacting with an asymmetrical deposit at different offset heights, seventeen experimental cases were selected for the non-rotated deposit to be numerically simulated. In order to provide a more direct comparison between the numerical and experimental results, the experimental results were replotted only showing the numerically selected cases, as seen in Figure 4.20.

![Figure 4.20. Comparison between experimental and numerical results for a non-rotated nozzle position.](image)
When qualitatively comparing the two results, it is evident that the gross physics of the supersonic jet interacting with an asymmetrical deposit are numerically captured. At a distance smaller than 15 mm from the surface of the deposit, at offsets larger than 2.38 mm in either the positive or negative direction, the turbulent nature of the jet is far too large for the jet to adhere to its surface, and the supersonic jet deflects instead, as seen in Figure 4.21(a). This explains the reduced drag force in comparison to the offsets between (2.38) mm and 2.38 mm, and is observed both experimentally and numerically. As the deposit is moved further away from the nozzle exit at these large offsets, the Coanda effect creates a low-pressure region at the rear of the deposit, which increases the drag force further compared to the drag predicted at offsets between (2.38) mm and 2.38 mm, as seen in Figure 4.21(b). Similar to the experimental results, when the jet is moved far enough away, the effect of turbulence is weakened, and the predicted drag force is comparable to other offset heights.

**Figure 4.21.** Velocity contours of jet interacting with an asymmetrical deposit for non-rotated nozzle position at a 7.13 mm offset; (a) 15 mm distance from surface of deposit; (b) 44 mm distance from surface of deposit.

Although, the qualitative comparison between the experimental and numerical results show similar trends, Figure 4.20 shows two main differences. The first is the lack of agreement between the measured and predicted drag force at an offset of (6.81) mm, and the second is that the Coanda
effect takes place at 15 mm away from the surface numerically, rather than 22 mm away from the surface as predicted experimentally. These discrepancies can be better explained by quantitatively comparing the experimental and numerical results.

In order to quantitively compare the experimental and numerical results, the percent difference between the two results is shown in Figure 4.22. As seen in the Figure, the percent difference is roughly 1-10% for most of the compared cases.

![Figure 4.22. Percent difference between experimental and numerical results.](image)

As the jet is moved further away from the center of the rotary shaft the percent difference increases; particularly when the jet is offset by (6.81) mm from the center of the rotary shaft. This result suggests that the supersonic jet leaving the nozzle may not be symmetrical, and could explain the increase in the percent difference as the nozzle is moved in either offset direction from the center. The following section will investigate the potential asymmetry of the jet.
4.3.3 Investigating Jet Asymmetry

Although care was taken in designing the experimental apparatus, it was hypothesized that the supersonic jet leaving the nozzle was not symmetrical. This was inferred from the increased percent difference as the jet was moved further away from the center of the rotary shaft, especially at an offset greater than 2.38 mm in either direction. Using a combination of experimental and numerical simulations, three potential causes were investigated: misalignment of the nozzle due to the brass pipe sections, the bend in the flexible hose from the air receiver, and internal pipe imperfections. These potential causes will be discussed in greater detail in the following sections.

Nozzle Position. There are multiple brass pipe sections leading from the flexible hose to the inlet of the nozzle. These pipe sections allow the connection of the shut-off valve, solenoid valve, and nozzle. However, if any of these pipe connections are misaligned, they could produce an asymmetrical jet at the nozzle exit. Using a vernier caliper, and the table on which the experimental apparatus rests as a reference, the overall pipe misalignment was calculated. It was found that the nozzle position was misaligned by 0.3 degrees. Similarly, since the vice assembly rests on a platform fabricated out of wood, it was also found to be misaligned by 0.2 degrees. As a way to investigate all of the associated misalignments with the pipe sections and vice assembly platform, the
asymmetrical deposit was removed from the rotary shaft, and a flat thin plate was 3D printed and placed in line with the centerline axis of the jet as seen in Figure 4.23. In this way, the torsion sensor could be used to locate the true zero position of the jet, since this flat plate has a large moment arm in comparison to the asymmetrical deposit, and is not as sensitive to the jet offset. With the flat plate placed on the rotary shaft, the jet impinged at perpendicular offsets that passed through the center of the rotary shaft until the torsion sensor recorded a moment of zero representing the jet true center position. At a distance of 66 mm away from the surface of the flat plate with respect to the centerline axis of the jet, it was found that the jet true center position was 1.2 mm below the originally anticipated center position of the rotary shaft. This validated the hypothesis that the jet leaving the nozzle exit is indeed not symmetrical and explains the lack of agreement between the experimental and numerical results for a non-rotated deposit position. The fact that the true center position is below the deposit further explains the exaggerated disagreement when the jet impinged above the deposit at an offset of (6.81) mm. Since the nozzle position needed to be offset by 1.2 mm below the center of the rotary shaft to compensate for the asymmetry of the jet, this would mean that the jet was actually impinging at an offset of (8.01) mm rather than the expected offset of (6.81) mm. Given the fact that an offset of (6.81) mm is a tangential position, it is clear that the jet was not adequately impinging on the deposit at this offset, and explains why the magnitude of the measured drag force was not similar to that observed for a 7.13 mm offset as predicted numerically. On the other hand, when the jet impinged below the deposit at an offset of 7.13 mm, the asymmetry of the jet caused the impingement to occur at an offset of 5.93 mm rather than the expected offset of 7.13 mm. In summary, the asymmetry of the jet caused the impingement to miss the deposit at offsets above the tangential position of the deposit, while at offsets below
the tangential position of the deposit, the jet impinged on the deposit but at slightly inaccurate offsets.

As a way to compensate for this asymmetry, the true zero position of the jet, which was determined to be 1.2 mm below the center position of the rotary shaft at a distance of 66 mm away from the surface of the flat plate was considered as the new center position in an experimental trial. The asymmetrical deposit was placed back onto the rotary shaft at a non-rotated position while maintaining this same distance from the nozzle exit to the surface of the rotary shaft. With this new center position, the jet impinged on the deposit at equal tangential offset heights above and below the deposit. If these misalignments were the primary cause of the jet asymmetry, then at a non-rotated position, the drag force should be equal above and below the deposit. However, this was not the case, suggesting that there are still other factors at play contributing to the asymmetry of the jet.

**Flexible hose connection.** As seen in the experimental design, there is a flexible hose that connects the air receiver to the main brass pipe section discussed above. It was hypothesized that the bend in the flexible hose, in conjunction with a relatively short straight pipe section, did not allow the high-pressure air to become fully developed. This undeveloped flow might lead to the jet asymmetry. In order to investigate the potential cause of this, the flexible hose along with the main pipe section was modeled using CFD, and a free jet case was simulated. Figure 4.24 shows the final mesh and the resulting velocity contour for this solution. This result clearly shows that the flexible hose connection is not the cause of the asymmetry.
Figure 4.24. Modeling flexible hose connection; (a) experimental image; (b) modeled fluid domain; (c) mesh design of pipe bend; (d) velocity contour of nozzle domain.
Pipe imperfections. Since the flexible hose connection was not the culprit in producing an asymmetrical jet, the possibility of imperfections within the pipe assembly causing obstructions to the flow of the jet was investigated. It was anticipated that if imperfections exist within the pipe assembly, this may generate an unbalanced turbulence that could lead to the jet asymmetry. Using CFD, there were two types of pipe imperfections modeled within the pipe assembly: spherical defects (Figure 4.25) and cubical defects (Figure 4.26). The cubical defects were offset at an angle of 45° to the flow of the jet, to create recirculating regions which further increase the turbulence of the jet leading to the inlet of the nozzle. The result of this work clearly shows that at supersonic speeds, the flow of the jet at the exit of the nozzle is not affected by these pipe imperfections, and that these are not the cause of the jet asymmetry.

In summary, this work demonstrated that the supersonic jet leaving the nozzle is not symmetrical, which explains the discrepancy observed between the experimental and numerical results for the asymmetrical deposit at a non-rotated position as the jet was offset further from the center of the rotary shaft. Based on the results of the investigative work, it is not entirely clear what causes all of the asymmetry; however, the discrepancy does not seem to be the result of an experimental design flaw.
Figure 4.25. Modeling spherical imperfections; (a) fluid domain; (b) mesh design of imperfections; (c) velocity contour of simulation result.

Figure 4.26. Modeling cubical imperfection; (a) fluid domain; (b) mesh design of imperfections; (c) velocity contour of simulation result.
CHAPTER 5

SURFACE PRESSURES OF SOOTBLOWER JETS INTERACTING WITH SYMMETRICAL DEPOSIT

The ability of CFD to accurately predict surface pressures is of practical importance in terms of understanding both deposit removal and deposit failure mechanisms within recovery boilers. As a way to validate the accuracy of the predicted pressure distributions, a setup similar to the one described in Section 3.1 was used, and pressure sensing films were placed on the surface of model deposits (Figure 5.1). As the supersonic air jet impinged on the deposit, the pressure sensing film directly recorded the pressure distribution and was used as way to compare the experimental result to CFD simulations.
5.1 Experimental Procedure and Methodology

The pressure sensing film selected for this application was Fujifilm Prescale. This type of pressure sensing film is comprised of two separate sheets: a color developing sheet and a microcapsulating sheet. As a force is applied to the surface of the film, the microcapsules within the microcapsulating sheet rupture onto the color developing sheet, producing an instantaneous and permanent high-resolution image of the pressure variation across the contact surface (Figure 5.2 [35]). The microcapsules break in proportion to the pressure they experience. In other words, the stain density at any point on the color developing layer is related to the pressure acting at that point. This relationship can be used to quantitively relate color density with pressure (Figure 5.3) [35].
5.1.1 Pressure Sensing Film Type

Fujifilm manufactures films with a variety of pressures ranges, so that film sensitivity is optimal for a specific application. If the film range is not carefully considered, then the microcapsules may either be too sensitive and become fully saturated by the applied force, or vice versa. In either case, the pressure variation obtained by these films would yield a result that is not meaningful.

Figure 5.3. Relating film color density to pressure.

Figure 5.4. Estimating the projected area of the jet.
In order to select the appropriate pressure range for this application, an estimate of the expected surface pressure on a symmetrical deposit at a distance of 18 mm away from the nozzle was calculated using the relationship presented in Equation 5.1. The projected area of the jet was calculated using the modeling software (Autodesk Fusion 360) by estimating that the core of the jet leaving the nozzle deviates 5° from the nozzle exit, as seen in Figure 5.4. This method predicted an area of 0.144 in² and referring to Figure 4.6, the force the symmetrical deposit experiences at 18 mm away is 25.3 N (5.68 lb). Therefore, the expected surface pressure can be estimated as follows:

\[
Pressure = \frac{force}{area} = \frac{5.68 \text{ lb}}{0.144 \text{ in}^2} = 39.4 \text{ psi} \quad ...(5.1)
\]

This meant that an ultra-low-pressure film type was the most suitable for this application, since this falls within the range of 28-85 psi as specified by Fujifilm.

### 5.1.2 Calibration Method

As mentioned previously, the qualitative pressure distribution can be quantitively related to a pressure distribution by means of a calibration curve, since the force is directly proportional to the color density within the specified pressure range. As a means of calibrating the films and obtaining a relationship between pressure and color density, a uniaxial Sintech tensile testing machine was used to apply known loads on the surface of the films. In order to apply reasonable forces that exert pressures within the range specified by Fujifilm, a cone-shaped calibration platform was 3D printed with a base diameter of 10.08 mm as seen in Figure 5.5. The Sintech was used to apply forces between 3.5-10.5 lbs in order to exert pressures between 28-85 psi.
Since the pressure sensing films are extremely sensitive to eccentric loading that lead to uneven pressure distributions and directly impact the accuracy of the calibration, a housing for a steel ball bearing was designed on the surface of the calibration platform. This ensured that the cone was point loaded along the central axis, and that the force would be evenly distributed across the area of the base plate.

![3D Printed Cone: Ball Bearing Housing](image)

**Figure 5.5.** Fujifilm calibration method.

A compressive load was applied to the surface of the pressure sensing film at a compression speed of 0.07 mm/min in increments of 1.76 lbs. Figure 5.6 shows a series of calibration stains obtained from the tests. The sensitivity of the pressure sensing film can be seen in Figure 5.6. If the compression head is not well aligned with the bottom plate of the calibration platform, the pressure distribution obtained will be significantly impacted. Therefore, a visual inspection of the pressure sensing film was essential to select the films with

![Calibration Stains](image)

**Figure 5.6.** Calibration stains.
the least amount of pressure discontinuities. In the above Figure all of the above stains would be rejected except for Figure 5.6(d). This procedure was done five times at each of the load increments and using the MFC-7560DN scanner, these images were scanned at the highest resolution. As a way to compare the image intensities to the known pressure, each of these images was first converted to a grey scale image, with a pixel value between 0 and 255. A pixel value of 255 corresponds to a pure white image, and a pixel value of 0 is for a pure black image. The average pixel intensities were computed for each of the calibration stains and they were plotted against their known pressures, as seen in Figure 5.7.

Using the above correlation, a MatLAB script was written that converted the scanned experimental pressure distributions to a greyscale image, and then applied the corresponding relationship from Figure 5.7 to each pixel value, before displaying a colormap of the pressure distribution as illustrated in Figure 5.8.

![Figure 5.7. Ultra-low pressure sensing film calibration curve.](image-url)
With the development of the calibration procedure above, the pressure sensing film was used to obtain a pressure distribution as the supersonic jet impinged on the surface of the symmetrical deposit. As a way to reduce complexity, the supersonic jet was not perpendicularly offset from the center of the rotary shaft, but was only moved in the axial direction of the jet further from the surface. The film was cut into 53.5 mm x 34 mm rectangles and was carefully wrapped around the surface of the deposit, and taped on the backside as seen in Figure 5.9. The force required to wrap the film \( F_{\text{wrap}} \) was estimated as follows, which shows that the pressure imposed on the film \( P_{\text{wrap}} \) was insignificant to rupture the microcapsules.

\[
P_{\text{wrap}} = \frac{\text{force}}{\text{area}} = \frac{2 F_{\text{wrap}}}{(2 r_{\text{deposit}}) x L_{\text{film}}} = \frac{F_{\text{wrap}}}{r_{\text{deposit}} x L_{\text{film}}} = \frac{2.5 \text{ lbs}}{0.375 \text{ in} \times 2.109 \text{ in}} = 3.2 \text{ psi} \quad \text{(5.2)}
\]
Although a considerable effort was made to create a tightly sealed contact area between the film and the surface of the deposit, it was not possible to have it perfectly air tight. As a result, the films became over saturated as the impinging air propagated within this contact area and created aggressive vibrations. As a way to overcome this, clear cellophane was used to wrap the outside of the film to reduce air infiltration and vibration of the film.

5.2 Comparing Experimental Surface Pressures with Numerical Models

The pressure sensing film was used at distances of 22 mm, 44 mm, and 110 mm from the surface of the deposit, and were similarly modeled in CFD. Figure 5.10 shows the comparison between the experimental and simulated results.
Figure 5.10. Comparison of experimental surface pressures and numerical surface pressures; (a) 22 mm from surface of deposit; (b) 44 mm from surface of deposit; (c) 110 mm from surface of deposit.

When qualitatively comparing the surface pressures obtained experimentally with the numerically predicted surface pressures, it is clear that the overall physics predicted by CFD are
being captured. As the supersonic jet leaves the nozzle exit, it entrains the surrounding fluid causing the jet to spread at a rate dependent on the level of turbulence. The spreading profile of the jet means its impinging area increases further from the nozzle exit with respect to the centerline axis of the jet. This spreading rate of the supersonic jet can be observed both experimentally and numerically. When the jet is near to the surface of the deposit, the close proximity means the jet has not significantly spread and the observed core diameter is small. As the jet is moved further away from the surface of the deposit, the jet spreads more significantly, which results in an increased core diameter. In order to quantitively compare the spreading rate of the jet, the core diameters were measured both experimentally and numerically. A summary of the results is presented in Table 5.1.

**Table 5.1.** Comparing the experimentally measured core diameters with the numerically predicted core diameters.

<table>
<thead>
<tr>
<th>Distance from Surface of deposit</th>
<th>Experimental Core Diameter (mm)</th>
<th>Numerical Core Diameter (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>22 mm</td>
<td>10</td>
<td>9.98</td>
</tr>
<tr>
<td>44 mm</td>
<td>10</td>
<td>10.2</td>
</tr>
<tr>
<td>110 mm</td>
<td>19</td>
<td>17.7</td>
</tr>
</tbody>
</table>

When comparing the pressure distribution between the experimentally measured surface pressure with the numerically predicted surface pressures, it is evident that the jet core pressures are comparable at close distances to the surface but deviate significantly as the jet is moved further away. The increased core pressures observed experimentally can be reasonably attributed to the differences in solution states between the two comparisons. The CFD result is a steady state solution, while the pressure sensing film effectively measures the transient state of the supersonic
jet since it is in place before the jet first impinges, which can explain the increased core pressures observed. Another difference between the experimentally measured surface pressure and the numerically predicted surface pressure is that there are high pressure regions observed experimentally that are not observed in the CFD simulations. These differences illustrate the sensitivity of the pressure sensing film since the model deposits are not perfectly smooth. Any roughness on the surface of the model deposit creates these high stress regions which increase the local pressures observed.
CHAPTER 6

CONCLUSIONS AND RECOMMENDATIONS

The hostile environment inside of the recovery boiler makes it difficult to install sensors or even visualize the effectiveness of the sootblower. Therefore, the majority of the experimental work to understand the fundamentals of deposit removal by sootblowers has been done using lab-scale experiments. More recently, with the development of numerical modeling software, a computational fluid dynamics (CFD) software package has been used to study the characteristics of a sootblower jet as it interacts with boiler tubes and deposits. With assumptions being used in the numerical model to reduce computational time and complexity, the accuracy of these simulations was not previously known. The validation of these models to date has mainly focused on quantitively validating a free jet, which does not capture the complex nature of the supersonic jet interacting with tubes and deposits. These complex interactions have only been validated qualitatively by visually comparing the gross features of the CFD analysis to schlieren images.
The experimental work performed in this study, in contrast, has quantitatively validated the complex interaction of a supersonic jet interacting with boiler tubes and deposits by measuring the reaction forces of the jet as well as the pressures imposed on the surface of deposits.

6.1 Conclusions and Practical Implications

Measuring reaction forces. When comparing the experimentally measured reaction forces of jet impingement on a single boiler tube to the simulation results, it was evident that the numerical model accurately captured the physics of this interaction. However, when the jet impinged on a round symmetrical deposit which was 3D printed, the agreement weakened since the 3D printed deposit was rough in comparison to the smooth rotary shaft representing the boiler tube. The effects of surface roughness seemed to diminish when subsequent comparisons were made of interactions with similarly formed 3D printed deposits which were asymmetrical. These results suggest that the geometry of these deposits plays a role on the accuracy of the CFD simulations. When comparing the geometry between both deposits, it is clear that the asymmetrical deposit is more aerodynamic than the symmetrical deposit. As a result, when the jet impinged on the asymmetrical deposit, its oblong shape deflected the supersonic jet more effectively, translating to insignificant viscous forces and a reduced effect of surface roughness. On the other hand, the impingement of the supersonic jet with a symmetrical deposit is blunter, causing larger viscous forces leading to more dramatic inaccuracies when surface roughness is not considered.

When comparing the experimental and numerical results as the jets impinged on the asymmetrical deposit at different perpendicular offsets with respect to the centerline axis of the jet, it was clear that the numerical model accurately captured the overall physics of these complex interactions. The Coanda effect, which causes the jet to adhere to the surface of the deposit when
impinged tangentially to the surface, was observed both experimentally and numerically. When the jet impinged at these tangential offsets at a position close to the surface of the deposit, the turbulence of the jet was too large to adhere to its surface. However, as the jet was moved further away from the surface of the deposit, the Coanda effect caused the jet to adhere to its surface and resulted in an increase in drag force. On the other hand, when the jet was moved far enough away from the surface of the deposit, the effects of turbulence were weakened, and the drag force was comparable to that observed at other perpendicular offsets. The lack of agreement between the experimental and numerical results as the jet was perpendicularly offset further from the center of the rotary shaft was caused by a slight asymmetry of the jet at the nozzle exit. The investigative work into the potential causes were inclusive and there was no evidence indicating that it was related to an inherent design flaw.

**Measuring surface pressures.** The pressure distribution obtained experimentally when compared to the CFD predictions yielded good agreement. The differences in the magnitude of the surface pressure observed experimentally can be reasonably attributed to the differences in solution states between the two comparisons. The CFD result is a steady state solution while the pressure sensing film effectively measures the transient state of the supersonic jet, since it is placed at a time before the jet first impinges.

The main contribution of this experimental work is a quantitative validation of the accuracy of the numerical models used to characterize a sootblower jet as it interacts with boiler tubes and deposits. The insight into the simulation accuracy provides confidence in numerically modeling more realistic recovery boiler interactions. Furthermore, this work shows that not modeling the
surface roughness of the deposit may introduce prediction inaccuracies depending on the deposit geometry.

6.2 Recommendations for Future Work

The following are recommendations for future work related to sootblowing optimization.

1. As discussed previously, the jet leaving the nozzle is slightly asymmetrical which led to a poor comparison between the experimental and numerical results in select cases where the jet impinged on the deposit at perpendicular offsets further from the center of the rotary shaft. Although the investigative work showed a lack of evidence that the asymmetry is due to an experimental design flaw, it is recommended that the main pipe section, nozzle, and vice assembly platform be re-manufactured with tighter tolerances. Furthermore, to reduce the vibration experienced by the rotary shaft upon impingement and increase its rigidity, a thicker stainless-steel rotary shaft can be used.

2. As further validation of the numerical results, the complexity of modeling pits within deposits that mimic crack formation can be investigated. This can be achieved by 3D printing a deposit with a symmetrical pit; separated by two halves down the pit center. With an additional normal sensor placed in such a way as to measure the normal stresses between the two halves of the pit, the supersonic jet can impinge on the surface inducing normal stresses within the pit which can be experimentally measured. These conditions can be similarly modeled using CFD, and the normal stresses produced within the pit can be compared to the numerical results.
REFERENCES


APPENDICES
APPENDIX A

Calculating Moisture Produced by Makita Compressor

As the air is compressed, the temperature rises increasing the amount of moisture it can retain. Once the compression process is complete and the air cools, the moisture absorbed during the heating process can no longer be retained causing the moisture to condense out. This condensed air has a relative humidity of 100%. As a way to calculate the amount of condensation produced by the compressor, a control volume was drawn around it and the final conditions were considered (as seen in Figure A.1).

Figure A.1. Control volume around Makita compressor.
According to the 2013 ASHRAE fundamentals handbook [36], the amount of condensation produced by the compressor can be calculated using the following equation.

\[
q_{\text{cond}} = \frac{q_{\text{air}} (W_{gr,\text{out}} - W_{gr,\text{in}})}{58310(v_{da})}
\]

Where \(q_{\text{cond}}\) is the condensation produced in gallons per min; \(q_{\text{air}}\) is the air flow through the compressor in cubic feet per min; \(W_{gr}\) is the grains of water per pound of dry air; \(v_{da}\) is the specific volume of dry air in cubic feet per pound of dry air.

The grains of water per pound of dry air and the specific volume of dry air can be obtained from the psychrometric chart according to the conditions specified in Figure A.1. It was assumed that the air flowing the compressor was 300 gallons per min.

\[
q_{\text{air}} = 300 \text{ GPM};
\]

\[
W_{gr,\text{out}} = 110 \frac{\text{grains of water}}{\text{lb dry air}};
\]

\[
W_{gr,\text{in}} = 66 \frac{\text{grains of water}}{\text{lb dry air}};
\]

\[
v_{da} = 13.5 \frac{\text{ft}^3}{\text{lb dry air}};
\]

Therefore, the amount condensation produced by the compressor at these conditions is 0.02 GPM. If the tank takes 5 min to fill up, this would produce 0.08 gallons (0.32 L).
APPENDIX B

One-Dimensional Mach Number Calculation

The Mach number for the given convergent-divergent nozzle can be calculated using a one-dimensional gas relation. The equation is designed to provide an exit Mach number for a fully expanded jet. A fully expanded jet is produced when the nozzle exit pressure is equal to the pressure of the surroundings. The exit Mach number required to generate a fully expanded jet is dependent on the ratio of the nozzle cross-sectional areas and the specific heat capacity of the gas at constant pressure and volume ($\gamma = c_p/c_v$) [37].

$$\frac{A_e}{A_t} = \frac{1}{Ma_e} \left[ \frac{1 + \frac{1}{2}(\gamma - 1)Ma_e^2}{\frac{1}{2}(\gamma + 1)} \right]^{\frac{1}{2}(\gamma+1)(\gamma-1)}$$

Where $A_e$ is the exit cross-sectional area; $A_t$ is the throat cross-sectional area; $\gamma$ is the specific heat capacity; $Ma_e$ is the exit Mach number.

$$\frac{A_e}{A_t} = 2.684;$$

$$\gamma = \frac{c_p}{c_v} = 1.4;$$

Substituting these knowns and simplifying:

$$0.00457(Ma_e)^6 + 0.068862(Ma_e)^4 + 0.345556(Ma_e)^2 - 2.68441Ma_e + 0.578009 = 0;$$

Finding the roots of this equation using MatLAB, the following solutions are:

$$0.2217;$$
$$2.527;$$
$$0.921 \pm 4.1315i;$$
$$-2.2953 \pm 2.7054i;$$
Since the Mach number needs to be greater than 1 for supersonic flows and the solution needs to be a real number, the exit Mach number to produce a fully expanded supersonic jet is 2.53.
APPENDIX C

Calibrating Experimental Sensors

The weight of the calibrated weights along with the bracket, string, and carabiner was 1.058 kg.

Therefore, the force applied is:

\[ F_{\text{applied}} = \text{mass} \times \text{acceleration} = 1.058 \, \text{kg} \times 9.81 \, \frac{\text{N}}{\text{kg}} = 10.38 \, \text{N} \]
Using the free body in Figure B.1, the force that Normal Sensor 1 experiences can be calculated as follows:

\[ \Sigma M_2 = 10.38 (D - x) - F_1(D) = 0 \]

\[ F_1 = \frac{10.38(D - x)}{D} [N] \]

Where \( D \) and \( x \) are measured in meters. Similarly, the force applied directly produces a moment as well on the torsion sensor as seen in Figure B.2.

\[ Moment = 10.38 \cdot r_{rotary} [N] \]

Where \( r_{rotary} \) is measured in meters. These forces were used to relate the voltage reading with the applied load.
APPENDIX D

Simulating the Effects of Screws Clamping Deposits

The effect of the screws on the numerically predicted drag force was investigated. A similar set out of boundary conditions and mesh was created as described in Section 3.5 with the addition of the screw models on the back side of the deposits. As a way to simplify the numerical models, the threads at the bottom of the screws were not modeled. Figure D.1 and Figure D.2 show the mesh and resulting velocity contour, respectively.

![Figure D.1. Mesh design of screws.](image1)

![Figure D.2. Velocity contour of resulting simulation.](image2)

The predicted drag force for the above simulation was 20.1 N which was an increase of 1.1 N from the original prediction in Section 4.2.2.
APPENDIX E

Jet Forces Exerted on Asymmetrical Deposits
APPENDIX F

Numerical Models of 45º Deposit Rotation

**Figure F.1.** Comparing experimental and numerical results of interaction with a 45º rotated deposit at a (6.81) mm offset, 66 mm away from the surface.

<table>
<thead>
<tr>
<th>66 mm from Surface @ (6.81) mm Offset</th>
<th>Drag Force</th>
<th>Moment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental Result</td>
<td>11.5 N</td>
<td>0.0177 Nm</td>
</tr>
<tr>
<td>Numerical Result</td>
<td>17.0 N</td>
<td>0.0876 Nm</td>
</tr>
</tbody>
</table>

**Figure F.2.** Comparing experimental and numerical results of interaction with a 45º rotated deposit at a 7.5 mm offset, 44 mm away from the surface.

<table>
<thead>
<tr>
<th>44 mm from Surface @ 7.5 mm Offset</th>
<th>Drag Force</th>
<th>Moment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental Result</td>
<td>23.1 N</td>
<td>0.262 Nm</td>
</tr>
<tr>
<td>Numerical Result</td>
<td>20.2 N</td>
<td>0.286 Nm</td>
</tr>
</tbody>
</table>
As can be seen in the above results, a similar trend to a 0° deposit rotation is observed. As the jet is offset further from the center axis of the rotary shaft, the discrepancy between the experimental and numerical results increases similarly, particularly when the jet is offset above deposit. This is due to the jet's asymmetry.