Enhanced Boiling Heat Transfer by Submerged, Vibration Induced Jets

A Thesis
Presented to
The Academic Faculty
by
Steven W. Tillery

In Partial Fulfillment
of the Requirements for the Degree
Master of Science in Mechanical Engineering

Georgia Institute of Technology
May, 2004
Enhanced Boiling Heat Transfer by Submerged, Vibration Induced Jets

APPROVED:

Marc K. Smith, Advisor

Ari Glezer

S. Mostafa Ghaasian

April 9, 2004
Date Approved by Chairman
ACKNOWLEDGEMENTS

I would like to express my sincerest gratitude to all of those who assisted me in the development of this thesis. First and foremost, I would like to thank my advisors, Dr. Marc Smith and Dr. Ari Glezer. Their advice and insight have served as a constant source of support throughout the duration of this project. I would also like to thank Dr. Sam Heffington. From the very beginning of my stint as a graduate student, Dr. Heffington has always been more than willing to offer any assistance that I may have needed, and for that, I am very appreciative. I would also like to thank my additional committee member, Dr. Mostafa Ghiaasiaan for his contributions to this project.

Most importantly, I would like to express my sincere appreciation to my family whose constant support and encouragement have been indispensable. I would especially like to thank my fiancée Sarah Chambers. The overwhelming flood of encouragement that I have received from both Sarah and her family has been invaluable, and I would like to thank them from the very bottom of my heart.
<table>
<thead>
<tr>
<th>TABLE OF CONTENTS</th>
</tr>
</thead>
</table>

ACKNOWLEDGEMENTS iii

LIST OF TABLES vii

LIST OF FIGURES viii

NOMENCLATURE xi

SUMMARY xiv

1. INTRODUCTION 1
   1.1 Background 3
       1.1.1 Immersion Cooling 3
       1.1.2 The Cavitation Jet 3
       1.1.3 The Cavitation Jet Heat Transfer Cell 4

2. LITERATURE REVIEW 7
   2.1 Effects of Jet Impingement on a Flat Heated Surface 8
   2.2 Impinging Jet Effects on Critical Heat Flux (CHF) 14
   2.3 Disadvantages of Impingement Jet for Heat Dissipation 17
   2.4 Immersion Cooling 19
   2.5 Streaming Forced Convection 20

3. EXPERIMENTAL APPARATUS AND PROCEDURES 23
   3.1 Physical Properties of Piezoelectric Diaphragm 23
       3.1.1 Input Voltage, Holder Diameter, and Bath Temperature 23
       3.1.1.1 Procedure for determining diameter dependency 25
       3.1.1.2 Procedure for determining temperature dependency 26
       3.1.1.3 Procedure for determining voltage dependency 26
       3.1.2 Threshold Surface Velocity Determination 27
   3.2 Pool Boiling Test Apparatus 28
       3.2.1 Actuator Assembly 28
       3.2.2 Test Module Design 29
   3.3 Pool Boiling Test Procedures 34
       3.3.1 Optimum Height Determination 34
       3.3.2 Optimum Angle Determination 35
       3.3.3 Efficacy of Turbulent Jet for Heat Removal 36
       3.3.4 Reduced Pressure Analysis 37
<table>
<thead>
<tr>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.4</td>
</tr>
<tr>
<td>3.4.1</td>
</tr>
<tr>
<td>3.5</td>
</tr>
<tr>
<td>3.5.1</td>
</tr>
<tr>
<td>3.5.2</td>
</tr>
<tr>
<td>3.5.3</td>
</tr>
<tr>
<td>3.6</td>
</tr>
<tr>
<td>3.6.1</td>
</tr>
<tr>
<td>3.6.1.1</td>
</tr>
<tr>
<td>3.6.1.2</td>
</tr>
<tr>
<td>3.6.2</td>
</tr>
<tr>
<td>3.6.2.1</td>
</tr>
<tr>
<td>3.6.2.2</td>
</tr>
<tr>
<td>3.6.2.3</td>
</tr>
<tr>
<td>4.</td>
</tr>
<tr>
<td>4.1</td>
</tr>
<tr>
<td>4.1.1</td>
</tr>
<tr>
<td>4.1.2</td>
</tr>
<tr>
<td>4.2</td>
</tr>
<tr>
<td>4.2.1</td>
</tr>
<tr>
<td>4.2.2</td>
</tr>
<tr>
<td>4.2.3</td>
</tr>
<tr>
<td>4.2.4</td>
</tr>
<tr>
<td>4.3</td>
</tr>
<tr>
<td>4.3.1</td>
</tr>
<tr>
<td>4.3.1.1</td>
</tr>
</tbody>
</table>
4.3.2  Heat Removal Capability of Cavitation Jet Using Calibrated Heater
   4.3.2.1  Optimum height determination  72
   4.3.2.2  Effects of cell rotation on heat transfer results  73
   4.3.2.3  Effect of flowrate on heat transfer  75
4.3.3  Comparison of Cavitation Jet to Single Round Gas Jet  77

5.  REPRODUCIBILITY AND ERROR  81
   5.1  Error Analysis  81
   5.1.1  Standard Lab Equipment Error  81
   5.1.1.1  Error in pressure transducer measurements  82
   5.1.1.2  Error in surface displacement and acceleration calculations  82
   5.1.2  Thermal Test Vehicle  84
   5.1.3  High Heat Flux Surface  85
   5.1.3.1  Surface Temperature  85
   5.1.3.2  Heat Flux  89
   5.1.3.3  Error associated with thermocouple holes  90

5.2  Reproducibility  94

6.  CONCLUSIONS AND RECOMMENDATIONS  96
   6.1  Variables Determining Jet Formation  96
   6.2  Jet Impingement in a Quiescent Pool  97
   6.3  Jet Impingement in a Channel Flow Environment  99
   6.4  Jet Performance Compared to that of Traditional Round Jet  101
   6.5  Recommendations for Future Research  102

REFERENCES  105
LIST OF TABLES

Table 2.1  Critical Heat Flux and Temperature Rise in Various Phase-Changing Cooling Mechanisms  8

Table 5.1  Uncertainties of Standard Laboratory Equipment  81
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1.1</td>
<td>The boiling curve (solid line). Regimes: (a) natural convection, (b) nucleate boiling, (d) transition boiling, and (f) film boiling. Point (c) is the critical heat flux limit, and (e) is the minimum heat flux limit.</td>
<td>5</td>
</tr>
<tr>
<td>Figure 3.1</td>
<td>Experimental set-up used to quantify properties of piezoelectric diaphragm.</td>
<td>24</td>
</tr>
<tr>
<td>Figure 3.2</td>
<td>a) solid model of actuator assembly, b) cross-section of piezoelectric diaphragm holder.</td>
<td>28</td>
</tr>
<tr>
<td>Figure 3.3</td>
<td>a) solid model of shims bolted to actuator assembly, b) part drawing of shims used to adjust the height and angle of driver.</td>
<td>29</td>
</tr>
<tr>
<td>Figure 3.4</td>
<td>a) solid model of aquarium wall, b) part drawing of aquarium wall.</td>
<td>30</td>
</tr>
<tr>
<td>Figure 3.5</td>
<td>a) solid model of aquarium top, b) solid model of aquarium base.</td>
<td>31</td>
</tr>
<tr>
<td>Figure 3.6</td>
<td>Part drawing of test aquarium top.</td>
<td>32</td>
</tr>
<tr>
<td>Figure 3.7</td>
<td>Part drawing of test aquarium bottom.</td>
<td>32</td>
</tr>
<tr>
<td>Figure 3.8</td>
<td>Pool boiling test assembly.</td>
<td>33</td>
</tr>
<tr>
<td>Figure 3.9</td>
<td>Schematic of pool boiling experimental apparatus.</td>
<td>34</td>
</tr>
<tr>
<td>Figure 3.10</td>
<td>Test parameters for the impingement angle and separation distance optimization tests.</td>
<td>36</td>
</tr>
<tr>
<td>Figure 3.11</td>
<td>Solid model of channel boiling test cell.</td>
<td>39</td>
</tr>
<tr>
<td>Figure 3.12a</td>
<td>Part drawing of side of cavitation heat transfer cell.</td>
<td>40</td>
</tr>
<tr>
<td>Figure 3.12b</td>
<td>Part drawing of bottom of cavitation test cell.</td>
<td>40</td>
</tr>
<tr>
<td>Figure 3.13</td>
<td>Photograph of channel flow heat transfer cell.</td>
<td>41</td>
</tr>
</tbody>
</table>
Figure 3.14 Schematic of channel boiling experimental apparatus. 42
Figure 3.15 Schematic of high heat flux assembly. 44
Figure 3.16 Detail of copper heater. 46
Figure 3.17 Detail of measurement region of heater. 47
Figure 4.1 Dye visualization photograph of the synthetic jet. The dye is entrained radially from an injection port near the edge of the diaphragm. 54
Figure 4.2 Threshold surface velocity required for jet formation. 55
Figure 4.3 Influence of bath temperature on the surface velocity at the resonant frequency of the vibrating diaphragm. 56
Figure 4.4 Effect of the inner diameter of the diaphragm holder on resonant frequency. Hole diameter = 17 mm (x), 15 mm (o), and 13 mm (△). 58
Figure 4.5 Effect of input voltage on the surface velocity of the oscillating diaphragm. 59
Figure 4.6 Effect of separation distance between jet driver and thermal test vehicle. Separation distance = 1.9 mm (●), 5.7 mm (x), 9.5 mm (o), 13.3 mm (△), 17.1 mm (□), and 20.9 mm (■). 61
Figure 4.7 Effect of impingement angle. Jet impingement angle with respect to horizontal = 0° (●), 15° (x), 25° (o), and 35° (△). 63
Figure 4.8 Effect of jet on (o) vs. jet off (□) on the heat dissipation from the thermal test vehicle. 64
Figure 4.9 Photographs of cavitation jet induced bubble ejection. 66
Figure 4.10 a) Visualization of low pressure dome at a pressure of 0.34 bar b) Visualization of ‘foaming’ phenomenon at a pressure of 0.14 bar. 67
LIST OF FIGURES Cont'd....

Figure 4.11 Effect of jet on (○) vs. jet off (×) on the heat dissipation from the thermal test vehicle in a channel flow environment.

Figure 4.12 Effect of jet when inlet water temperature is 20°C (○) vs. effect of jet when inlet water temperature is 60°C (△).

Figure 4.13 The effect of the separation distance on heat transfer enhancement. Separation distances: 2.5 mm (●), 5 mm (○), 7.5 mm (▲), 10 mm (●).

Figure 4.14 The effect of inclining the cavitation jet heat transfer module on the point where the jet no longer affects the amount of heat dissipated from the calibrated copper heater. Baseline case with no cavitation jet (●), jet actuated with no incline (▲), jet actuated with cell inclined 45° (×).

Figure 4.15 Heat dissipation capabilities of channel boiling heat transfer cell with the cavitation jet activated and a crossflow rate of: 0 ml/s (●), 1.5 ml/s (○), 3.5 ml/s (▲), 5.5 ml/s (●), and 7.5 (×).

Figure 4.16 Variation of the inverse of centerline velocity (○) and jet half-width radius (△) with distance from diaphragm.

Figure 4.17 $yU_{cl}$ vs. $y$

Figure 5.1 Reproducibility of cavitation jet for heat removal in channel boiling heat transfer cell.
NOMENCLATURE

$q$  heat flux (time-averaged value), $(W/m^2)$
$q_{crf}$  critical heat flux, $(W/m^2)$
$U$  velocity of liquid jet, $(m/s)$
$h_f$  latent heat of evaporation, $(J/kg)$
$D$  diameter of heated surface
$c_{pl}$  specific heat of liquid at constant pressure, $(W/m^3K)$
$T_{sat}$  saturation temperature, $(K)$
$T_{eq}$  saturation temperature, $(K)$
$We$  Weber number
$Nu$  Nusselt number
$H$  constant heat flux
$S$  based on spacing
$L$  length
$Ra_{"}$  flux based Rayleigh number
$\text{sub}$  subcooling
$V_{p-p}$  peak-to-peak velocity, $(mm/s)$
$V_{rms}$  voltage, $(V)$
$f$  frequency, $(Hz)$
$A$  surface acceleration, $(mm/s^2)$
$u$  uncertainty
$\text{trans}$  transducer
NOMENCLATURE Cont’d….

$x$ correlation factor for vibrometer, (mm/s/V)

$mult$ multimeter

$press$ pressure

$T$ temperature

$diode$ diode

$DAS$ data-acquisition system

$I$ current, (A)

$z$ distance, (mm)

$s$ surface

$A_{TC}$ area of neck, (mm$^2$)

$A_s$ area of surface, (mm$^2$)

$wire$ thermocouple wire

$TC$ thermocouple

$k$ thermal conductivity, (W/mK)

$meas$ measured

$holes$ thermocouple holes

$solid$ heat with no thermocouple holes

$l$ depth of thermocouple holes, (mm)

$r$ radius, (mm)

$J$ momentum flux, (kg-m/s$^2$)

$X$ distance between point in jet and virtual origin, (mm)
NOMENCLATURE Cont’d....

$U_{cl}$  centerline velocity of jet, (m/s)
$d$  diameter of jet origin, (mm)
$V_e$  exit velocity of jet from virtual origin, (cm/s)
$R$  radius of heated surface, (mm)
$Pr$  Prandtl number
$Z$  separation distance, (mm)

Greek

$\rho_l$  density of liquid, (kg/m$^3$)
$\rho_v$  density of vapor, (kg/m$^3$)
$\varepsilon_{sub}$  correction factor for the effect of subcooling on critical heat flux
$\sigma$  surface tension, (N/m)
$\delta$  surface displacement (mm)
$\omega$  frequency, (rad/s)
$\alpha$  error
$\nu$  kinematic viscosity, (m$^2$/s)
SUMMARY

In this analysis, the efficacy of the cavitation jet for heat transfer enhancement was demonstrated. The cavitation jet was formed from a cluster of cavitation bubbles that are the result of a submerged piezoelectric diaphragm oscillating above a given velocity threshold. The cavitation jet was positioned in such a way as to impinge a submerged heated surface. By impinging the heated surface, the vapor bubbles that would normally cling to the surface during nucleate boiling were removed from the surface and were replaced with cool water from the bulk within the test module. In this analysis, two different heaters operating in two different flow environments were examined. For each heater in each environment, the cavitation jet significantly increased the heat transfer from the heated surface.

There are many factors that dictate how and when the cavitation jet is formed. First and foremost, to generate a jet from the cavitation bubbles formed on the surface of a submerged piezoelectric diaphragm, the vibrating surface must exceed a threshold surface velocity. The threshold velocity was found to be 1100 mm/s. To achieve this threshold, there are many factors that must be considered.

Through this analysis, it was demonstrated how the resonant frequency of the diaphragm increases when the inner diameter of the holder onto which it is mounted decreases. It was determined that below an inner diameter of approximately 14 mm, the diaphragm can no longer achieve the surface velocity necessary for jet formation at its respective resonant frequency. In addition to the holder’s inner diameter, another key variable is the temperature of the water bath into which the jet is submerged. As the
temperature of the bath was increased, the surface velocity, for a given frequency and
input voltage, decreased. It was shown that when the temperature of the driver is above
50°C, the surface can no longer achieve the surface velocity necessary to create a jet since
the oscillating diaphragm departs from resonance at the increased temperature.

The first set of experimentation was conducted in a quiescent pool environment.
By impinging a submerged heated surface with the cavitation jet in this environment, heat
fluxes of up to 135 W/cm² were achieved in a compact geometry requiring approximately
1 W of input power. This is an improvement of approximately 230% when compared to
traditional pool boiling at the same surface temperature. With the jet activated, the
convective heat transfer coefficient is approximately 1.29 W/cm²K compared to 0.4
W/cm²K with the jet off. It was found that the optimal separation distance between the
diaphragm and the heated surface is approximately 5.7 mm. Comparative tests using
different impingement angles indicate relatively no difference in the heat flux dissipation
as a function of angle.

Reduced pressure analyses indicate that creating a cavitation jet is possible at
pressures as low as 0.34 bar. Below this pressure, bulk fluid motion continues but not in
a concentrated form. This bulk fluid motion appeared to be a result of a large low
pressure region located directly below the vibrating diaphragm. This low pressure region
formed a dome of cavitation bubbles beneath the diaphragm’s surface. At approximately
0.14 bar, this dome region began growing rapidly as the entire test module filled with
bubbles. All of the noticeable bulk fluid motion ended when the pressure in the chamber
reached 0.05 bar.
In addition to the pool boiling experimentation, a channel boiling heat transfer cell was constructed such that the efficacy of the jet for removing heat in a flow field could be examined. In this experimentation, the cavitation jet was used to remove the vapor bubbles that formed on two different heaters during nucleate boiling and force them back into the cooler bulk liquid where the crossflow within the cell swept them downstream. For a surface temperature of 120°C, heat fluxes up to 165 W/cm² were achieved in a compact geometry requiring approximately 1 W of input power. This is a heat flux increase of 236% compared to traditional channel boiling with no jet, for the same surface temperature. At a bath temperature of 20°C with the jet actuated, the convective heat transfer coefficient increases from 0.4 W/cm²K to 1.33 W/cm²K. It was also determined that by increasing the temperature of the water flowing through the cell from 20°C to 60°C, the percent improvement in heat dissipation resulting from impingement by the cavitation jet increased to 300%. When the bath temperature is 60°C and the jet is actuated, the heat transfer coefficient is 1.87 W/cm²K at a surface temperature of 115°C. This is a 29% improvement over the case where the water was 20°C. This increase in heat transfer coefficient is a direct result of the increased conductivity of the water at the higher temperature.

Similar to the pool boiling experimentation, the optimal separation distance between the surface of the driver and the heated surface is approximately 5 mm. The major difference that was noticed with the induced crossflow was the fact that the jet’s performance did not degrade at separation distances on the order of 1 to 2 mm as it did in the pool boiling experimentation. In the case of pool boiling, the performance of the
cavitation jet declined in close proximity of the heated surface due to the heated water and vapor becoming trapped between the driver and the heated surface

In addition to the separation distance, another important parameter that was analyzed was the flowrate of water through the cell. With no flow, the confined geometry of the cell caused the water within the cell to heat to approximately 70°C when approximately 100 V was supplied to the heater. This led to critical heat flux, 240 W/cm², being reached at a surface temperature of 140°C. By supplying even a minimal amount of flow through the cell, critical heat flux was not reached before the capacity of the experimental apparatus was reached. In this experimentation, the lowest flowrate tested was approximately 1.5 ml/s, which caused the average water temperature within the cell to be approximately 40°C at 100 V of input power. At this flowrate, the boiling curve was very similar to that which occurs with no flow with the major difference being that critical heat flux was not reached. It was found that by increasing the flowrate, the heat transfer improvement capabilities of the jet declined. Based on the results of this analysis, the best possible flowrate is the smallest one tested or 1.5 ml/s.

When using the calibrated heater as the heat source, it was shown that the greatest amount of improvement in heat flux for a given surface temperature occurred at 120°C. Beyond this point, the effectiveness of the jet declined until the surface temperature of the heater reached 150°C at which the amount of heat dissipated from the surface was approximately 300 W/cm². At this point, the jet no longer had any effect on the heat transfer from the calibrated heater. By inclining the heat transfer module and calibrated heater 45° such that the water inlet was positioned below the water outlet, the influence of the cavitation jet did not become minimal until a much higher surface temperature was
achieved. Because of this, a heat flux of over 350 W/cm² was attainable before the jet no longer influenced the heat dissipation from the surface of the heater.

To obtain the Reynolds number of the cavitation jet used in this analysis, a similarity solution was utilized. After calculating the virtual origin of the cavitation jet, the Reynolds number was found to be 1450. In order to quantify the effectiveness of the cavitation jet, its Nusselt number was compared to that of a traditional turbulent round jet. Using the parameters established from the similarity solution coupled with the heat transfer coefficient obtained for pool boiling, the experimental Nusselt number of the cavitation jet was found to be 42. Using an experimentally derived correlation, the Nusselt number of a traditional turbulent round jet was found to be 53. By comparing the Nusselt numbers of the cavitation jet with the one obtained from the correlation for the traditional round jet, it was determined that the cavitation jet provides less heat transfer improvement than a traditional round turbulent jet operating under the same parameters.
CHAPTER 1

INTRODUCTION

As a result of the numerous technological advances in microelectronic circuit design, component size is steadily decreasing and at the same time, performance demands are increasing. As electronic packages increase in speed and capability, the heat flux that must be dissipated to maintain reasonable chip temperatures has also risen. Since the frequency of component failures dramatically increases at elevated temperature levels, research efforts in recent years have been focused on developing methods to adequately remove heat from the package thus allowing it to operate at acceptable temperature levels.

Even though the heat transfer rate from most electronic chips is rather modest, the surface area of the chip is usually very small resulting in heat flux values up to $10^6$ W/cm$^2$. As a result, electronic cooling techniques differ from more traditional methods of heat transfer due to the need to operate in confined areas and maintain low operating temperatures, while maintaining predictable temperature fluctuations. Past methods of cooling components have simply used natural or low-speed forced air convection. These heat transfer mechanisms are effective in providing heat fluxes of approximately 5 to 10 W/cm$^2$ [1]. As a result, more creative techniques are needed in order to dissipate the large heat fluxes that will be generated by future electronic devices.

Two-phase heat transfer can provide the large heat fluxes needed for microelectronic packages to operate at acceptable temperature levels. By changing the
phase of the working fluid, a two-phase heat transfer cooling scheme supports high heat transfer rates across moderately small temperature differences. For example, direct nucleate boiling on a heated surface has been shown to be a very effective technique for dissipating a large amount of heat at moderate temperature differences with small fluctuations in surface superheat [2].

A subject of research in recent years at Georgia Tech has been the development of a turbulent jet formed from the collapse of the cavitation bubbles that form on the surface of an oscillating diaphragm. Cavitation is defined as the formation of an air or vapor pocket (or bubble) due to lowering pressure in a liquid, often as a result of the collapse of a vapor bubble [3]. In the case of the cavitation driven jet, this low-pressure region forms under the surface of a submerged oscillating piezoelectric diaphragm vibrating at resonance.

Immersion cooling is the act of submerging a heated surface into a fluid in order to promote convective heat transfer from the surface to the fluid. A cooling module based on a cavitation jet capitalizes on the benefits of two-phase immersion cooling while improving traditional pool boiling heat transfer. Direct impingement by a cavitation jet onto a heated surface not only forcibly removes the vapor bubbles that cling to the surface as critical heat flux is approached, it also creates a recirculating flow which rewets the surface with cool bulk fluid. By coupling these two heat transfer enhancement techniques into a single heat transfer module, the onset of critical heat flux was shown through experimentation to be substantially delayed. Because of this, a heat transfer module based on the cavitation jet technology has the potential to dissipate more energy for a given surface temperature than many other immersion coolers.
1.1 Background

1.1.1 Immersion Cooling

Immersion cooling is a highly effective cooling strategy for two reasons: 1) the high thermal conductivity of the liquid medium as opposed to that of air enhances natural or forced convection from the heated surface, and 2) the liquid to vapor phase change at higher heat fluxes is an extremely effective means for heat removal. Recently, interest in liquid immersion coolers has increased because of the need for effective cooling in microelectronic devices [4]. The performance of an immersion cooler at the high heat fluxes required of present and future applications is possible because of the nucleate boiling that occurs with direct contact on the heated surface [5]. A key reason for the efficient heat transfer that occurs during boiling is that buoyancy forces (i.e., gravity) remove the vapor bubbles generated at the heated surface. When the heat flux from the surface is increased past a critical level, a large, possibly catastrophic increase in temperature occurs. This critical heat flux marks the transition from the nucleate boiling to film boiling. In film boiling, a thin insulating layer of vapor completely covers the heated surface, which then produces the large temperature increase. This transition occurs at much lower heat fluxes in a microgravity environment because buoyancy forces are almost negligible. Thus, the performance of immersion cooling in this environment is drastically reduced.

1.1.2 The Cavitation Jet

The cavitation jet is formed from the cavitation bubbles located at the center of an oscillating piezoelectric diaphragm submerged in water. As proposed by James et. al.
[6], the diaphragm-driven turbulent water jet has finite linear streamwise momentum, but in contrast to conventional jets, it is formed without net mass injection across the flow boundary and is comprised entirely of axisymmetrically entrained fluid. After the oscillating frequency of the diaphragm surpasses a given threshold, the jet appears near the center of the diaphragm along with a small cluster of cavitation bubbles. Pressure oscillations due to vortex ring coalescence in the liquid near the surface of the diaphragm result in the time-periodic formation and collapse of these cavitation bubbles, which entrains surrounding liquid and generates the turbulent jet directed normal to the surface of the diaphragm and flowing away from its center. Even though the jet results from time-periodic excitation, the time-averaged jet structure is similar to that of a conventional turbulent round jet. Both the width and centerline velocity of the jet are linear functions of the distance from the diaphragm.

According to James, the jet appears to become turbulent at or near the surface of the diaphragm and it spreads linearly with streamwise distance. However, the jet Reynolds number is approximately 1449, based on a jet diameter equal to the characteristic diameter of the cavitation bubbles (approximately 2 mm) and a streamwise velocity of approximately 65.8 cm/s. Nevertheless, the mean flow of the jet exhibits a remarkable tendency toward self-similarity as has been extensively documented for high Reynolds number conventional jets (e.g., Rajaratnam [7]).

1.1.3 The Cavitation Jet Heat Transfer Cell

By utilizing a cavitation jet to enhance two-phase heat transfer, it was conjectured that the heat dissipation requirements of future electronic packages could be attained. In
order to test this presumption, two different heat transfer cells were constructed whose collective purpose was to test the efficacy of the cavitation jet for improving heat transfer in both a quiescent pool as well as in a channel flow environment. In each of these experiments, the cavitation jet was positioned such that it impinged onto a heated surface. The impact of the cavitation jet effectively removed from the heated surface the bubbles which formed during nucleate boiling, while rewetting the surface with cooler bulk fluid.

To describe the efficacy of the cavitation jet for improving heat transfer, consider the standard boiling curve shown schematically as the solid line in Figure 1.1. This curve is a plot of the heat flux from a horizontal solid surface completely covered by a pool of liquid versus the wall superheat (the wall temperature minus the saturation temperature). Various heat transfer regimes are marked on the plot by lower-case letters. The
beginning of the curve is Regime (a), where the heat transfer from the surface is due to natural convection. The next part of the curve is Regime (b), where nucleate boiling occurs. Here, vapor bubbles form at specific nucleation sites on the heated surface. They grow until they reach a large enough size for the buoyancy force to remove them from the surface. As the heat flux increases, the type of nucleate boiling changes from one of single bubble growth and detachment to a fully-developed regime in which vapor is removed from the surface in the form of plumes or jets. The plumes from several adjacent nucleation sites will also merge above the surface to form much larger vapor bubbles. As the heat flux is increased a bit more, the rate of vapor generation exceeds the rate at which vapor is effectively removed from the surface. This is the critical heat flux limit, Point (c). When the heat flux is increased above this limit the system undergoes a transition to film boiling, Regime (f). Here, a thin vapor film completely covers the solid surface. The vapor insulates the surface and so the wall superheat is dramatically increased as shown. Such high temperatures may damage sensitive equipment and is usually avoided during the operation of immersion coolers. Transition boiling, Regime (e), is a state in which an intact vapor film covering the solid surface is unstable. Finally, the boiling curve displays a hysteresis effect. Once film boiling occurs, it is stable at heat fluxes below the critical heat flux limit. Only when Point (e), the minimum heat flux limit is reached does an intact vapor film become unstable. Below this point, the system returns to a stable state of nucleate boiling.
CHAPTER 2

LITERATURE REVIEW

Over the last 50 years, jet impingement by single or multiple jets has been a key area of study due to its ability to remove large amounts of heat from a surface with minimal input power. Comprehensive reviews of this topic have been presented by many authors including Downs and James [8] and Hrycak [9]. Recently, increased attention has been directed to applications of liquid jets as the heat transfer coefficient can be increased several orders of magnitude in comparison with that of gas jets [10]. As an attractive means of providing very high heat/mass transfer rates, liquid jet impingement has been employed in cooling systems of heat engines [11] or electronic devices [12] and thermal treatments of metals [13]. With liquid as the working fluid, two operation modes are possible: submerged jet and free-surface jet. In the former case, a liquid jet is discharged into the same liquid. In the latter, a liquid jet is exposed to a gaseous environment [14].

Because of the importance of understanding the physics behind both heat and mass transfer in many common industrial applications, Martin et al. [15] composed a comprehensive survey emphasizing the engineering applications of impinging jets rather than a basic theoretical approach. Therefore, the work by Martin provides many empirical equations that can be used to predict heat and mass transfer coefficients from
round nozzles, arrays of round nozzles, single slot nozzles, and arrays of single slot nozzles.

The phenomenon of critical heat flux has been studied in great detail and semi-theoretical correlations for the critical heat flux have been developed by many authors [16]. Table 2.1 displays the critical heat flux and temperature rise in various commonly studied two-phase cooling mechanisms [17].

<table>
<thead>
<tr>
<th>Cooling Type</th>
<th>CHF (kW/cm²)</th>
<th>ΔT (Wall-Liquid)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spray Cooling</td>
<td>0.6</td>
<td>Small</td>
</tr>
<tr>
<td>Jet Boiling Cooling on Micro-Surface</td>
<td>0.5</td>
<td>Large</td>
</tr>
<tr>
<td>Jet Boiling Cooling on Macro-Surface</td>
<td>0.1-0.3</td>
<td>Large</td>
</tr>
<tr>
<td>Subcooling Convective Boiling</td>
<td>0.5</td>
<td>Large</td>
</tr>
<tr>
<td>Mini-Channel Convective Boiling</td>
<td>0.2</td>
<td>Large</td>
</tr>
<tr>
<td>Micro-Channel Convective Boiling</td>
<td>0.3</td>
<td>Large</td>
</tr>
<tr>
<td>Pool Boiling</td>
<td>0.1</td>
<td>Large</td>
</tr>
<tr>
<td>Heat Pipe</td>
<td>0.01-0.03</td>
<td>Large</td>
</tr>
</tbody>
</table>

2.1 Effects of Jet Impingement on a Flat Heated Surface

To begin this overview of past research conducted in an effort to increase the amount of heat dissipated from a surface via an impinging jet, an analysis conducted by Garg et al. [18] will be discussed. The research conducted for this set of experimentation is similar to that conducted and discussed in this thesis in that the authors utilized
vibrating piezoelectric diaphragms to create liquid jets. In this study, synthetic jet enhancement of natural convection and pool boiling heat transfer in an enclosure, filled with a dielectric, electronic cooling liquid (FC-72) was studied. As previously mentioned, the jet was produced by a diaphragm-driven actuator that was operated at resonance and produced planar, submerged liquid jets that impinging upon a flat foil heater and spread laterally along its surface. The induced convective thermal transport from the heated surface yielded nearly a four-fold improvement in heat transfer over unassisted natural convection. The highest enhancement of pool boiling was observed near boiling incipience, with the synthetic jet producing an earlier transition to nucleate boiling than encountered in a quiescent pool. Substantial thermal transport enhancement was also attained in the pool boiling regime, with an increase by as much as a factor of three in the surface heat flux at a fixed heater superheat. Most significantly, the largest enhancements occurred at heat flux values in the transition region between natural convection and nucleate boiling, apparently due to earlier incipience and transition to nucleate boiling in the synthetic jet experiments.

An experimental investigation of nucleate boiling heat transfer from modified surfaces cooled by multiple in-line impinging circular jets was conducted by Kugler [19] and was found to agree with single jet results. A copper block was heated from the back by two electrical arcs, and cooled on the opposite side by three identical liquid jets of distilled water at subcoolings of 25°C, 50°C, and 77°C and Freon 113 at 24°C subcooling. Liquid flow rates were held constant at 5, 10, and 15-gph for each of the three jets with jet velocities ranging from 1.4 m/s to 11.2 m/s and jet diameters ranging from 0.95 mm to 2.2 mm. To increase the maximum heat flux and heat removal rates, the boiling surface
was modified with both macro and micro enhancements. Macro modification consisted of machined radial grooves in the boiling surface arranged in an optimally designed pattern to allow better liquid distribution along the surface. These grooves also reduced splashing of liquid droplets, and provided 'channels' to sweep away bubbles. Micro modification was achieved by flame spraying metal powder on the boiling surface, creating a porous, sintered surface. With the addition of both micro and macro structured enhancements, maximum heat flux and nucleate boiling were enhanced by more than 200%. Examination of each surface modification separately and together indicated that at lower superheats, the micro structure provided the enhanced heat transfer by providing more nucleation sites, while for higher superheats, the macro structure allowed better liquid distribution and bubble removal.

Experiments were performed to study the effect of nozzle rotation, flow rate, degrees of subcooling, number of jets, and velocity on the heat flux in jet impingement cooling using deionized distilled water [20]. In these experiments, an apparatus was constructed such that multiple water jets were mounted above a calibrated heater so that impingement by the jets would cause a film of water to constantly rewet the heated surface. In each of these experiments, the height of the film varied from 0.5-1.5 mm, and because the film was of the order of a millimeter in thickness, conduction through the film and subsequent evaporation at the liquid surface was insignificant. From this research, it was found that the heat flux increased appreciably with increase in flow rate with the highest measured in this experimentation being 598 W/cm² at a flow rate of 4.2 L/hr. No significant change in heat flux was measured by varying: jet velocity, the number of jets, or the rate of rotation, which was observed at any particular fixed flow.
rate. The authors theorize that a further increase in heat flux could be realized if efforts are directed towards maintaining a very thin film on the heated surface (less than 3 μm). This increase would be due to both the enhanced evaporative cooling by conduction as well as the reduction of resistance for the vapor to escape that an extremely thin film would pose.

An extensive study was conducted by Chang et al. [21] to evaluate heat transfer characteristics for single-jet and multi-jet impingement in submerged confined liquid flows as utilized in avionic heat transfer enhancement units. The experiments covered a large range of turbulent jet Reynolds numbers with different plate spacing-to-jet diameter and orifice pitch-to-jet diameter ratios.

Single-jet data were obtained for both stagnation point Nusselt numbers and local-average Nusselt numbers to permit prediction of the average heat transfer over a region extending outward from the stagnation point. It was observed that the stagnation point Nusselt number decreases slightly with an increasing plate spacing-to-jet diameter ratio, even at ratios that would still be within the potential core in unconfined flows, thus contradicting the results of others. The authors assumed that this difference is caused primarily by the recirculation vortex in a confined flow that contributes to the breakup of the emerging jet. Multi-jet experiments clearly showed the strong jet interaction between adjacent jets. These reduced the stagnation point heat transfer compared to a single jet, but slightly increased the heat transfer away from the stagnation point.

Rach and Holman [22] studied boiling heat transfer to a freon-113 jet impinging upward onto a flat, heated surface. Based on the results of their experimentation, they concluded that the heat transfer, heat flux and the area of contact during nucleate boiling
of an impinging liquid jet are all independent of the orientation of the heater surface over the range of 0 to 45 degrees relative to horizontal. They also found that nucleate boiling heat flux is further independent of the jet velocity and the inner diameter of the nozzle, being dependent solely on the saturation temperature excess. Thus the jet velocity and nozzle inside diameter only serve to increase the area of contact and, as a result, the heat transfer during nucleate boiling. Studies conducted in the film boiling regime concluded that the heat flux is independent of jet nozzle diameter and the test surface orientation, although the heat transfer and contact area are dependent on these variables.

An analysis was conducted by Ishigai et al. [23] to study the effects that an impinging water jet has on a high temperature flat surface when measured at the jet’s stagnation point. Through their experimentation, it was observed that the boiling curve for the impingement zone of a water jet is shifted to a higher heat flux and higher wall superheat by increasing the impinging jet’s velocity or the subcooling of the fluid. With a subcooling temperature of 55°C and a jet velocity of 2.1 m/s, the maximum amount of heat dissipation recorded was 1.5 x 10^3 W/m². The boiling curve obtained in this analysis is not always monotonous from the minimum heat flux to the maximum, but it instead has a ‘shoulder’ of nearly constant heat flux within a certain wall temperature range at some conditions. It was also noticed that in the film boiling region, no liquid-solid contact was observed, and the beginning of the contact coincides with the minimum heat flux point. The total heat flux of the film boiling obtained by the experiments is 1.6-1.7 times as high as the value calculated by a two-phase boundary layer theory analysis, but both tendencies are in agreement.
Ma et al. [24] performed an investigation to provide fundamental information concerning the heat transfer process of a modified technique of a spray cooling for microelectronic devices. Two-phase two-component jets of nitrogen gas and water droplets were impinged upon simulated micro-chips with high velocity under atmospheric pressure. Heat fluxes as high as $2.8 \times 10^6$ W/m$^2$ were dissipated from vertical chip-size heaters at a wall temperature of 58°C by means of the aforementioned cooling technique. It was determined in this analysis that the optimum spacing between the nozzle, from which the two-component jet originated, and the heated surface was 1.5 mm. One key outcome of this analysis was the fact that the heat transfer coefficient sharply decreased with the increasing of heat flux (for heat fluxes less than $9 \times 10^5$ W/m$^2$). Since the heat transfer coefficient is typically unaffected by the variation of heat flux for single-phase convection, this trend caused the authors to make the assumption that the heat transfer being witnessed in this analysis was neither single-phase convection nor nucleate boiling. It was hypothesized that the forced convective cooling was the source of this observed phenomenon.

Ma et al. [14] also performed an experimental study to investigate the effect of the jet inclination on convective heat transfer from vertical heaters to circular submerged liquid jets of large of large Prandtl number. Local heat transfer measurements were made with obliquely impinging circular jets of transformer oil issued from both pipe-type and orifice-type nozzles in the range of jet Reynolds number from 162 to 958, corresponding to the jet velocity from 1.73 to 19.1 m/s. Displacement of maximum heat transfer points was observed and measured with jet inclination angles from 90° to 45°. The authors found that displacement measured with submerged oil jets is slightly larger than that with
free-surface oil jets. Maximum heat transfer coefficients were measured and were found to decrease with increasing jet inclination. The measured maximum Nusselt numbers with the pipe-type nozzle are slightly higher than those measured with the orifice-type nozzle. The authors also measured the local heat transfer profiles along both the x and y-axis. The x-axis profiles exhibit an increasing asymmetry with increasing jet inclination. The y-axis profiles are essentially independent of the variation of the jet inclination. It was also found that local heat transfer profiles on both the x and y-axis are insensitive of the Reynolds number of the impinging jet.

2.2 Impinging Jet Effects on Critical Heat Flux (CHF)

Critical heat flux phenomenon was studied by Katto et al. [25] for a nucleate boiling system in which a saturated liquid (water, Freon-113, and trichlorethene) was supplied through a thin plane jet placed near the front edge of a heated rectangular surface. Critical heat flux takes place under the state of fully-developed nucleate boiling, in which nucleate boiling become so vigorous that the boiling heat transfer is no longer affected by the forced convection. In the fully-developed nucleate boiling at sufficiently high heat fluxes, part of the liquid supplied from the plane jet was splashed away from the heated surface by the vapor ejected from a layer of nucleate boiling liquid covering the heated surface. However, the layer of nucleate boiling liquid itself was maintained on the heated surface until critical heat flux took place. As for the plane jet, its incidence velocity alone was found to be the only important factor in determining the point at which critical heat flux occurs. Neither the thickness nor the incidence angle has an
effect on critical heat flux. Based on the results of their study, the authors conjectured that as far as critical heat flux is concerned, it seems likely that there are many affinities between the boiling system with a plane jet and one with an impinging round jet.

Nucleate boiling at very high heat fluxes was created on a heated surface covered with a flowing film of saturated water at atmospheric pressure being maintained by a small circular jet of water held at the center of the heated surface [26]. In this study, nucleate boiling of saturated water at atmospheric pressure was created at heat fluxes up to \(2 \times 10^4\) W/m\(^2\) on a heated 8 mm X 8 mm square surface with an impinging 2 mm diameter, circular jet. Boiling in this system seemed to appear as a monotonous extension of the boiling curve of ordinary high heat flux pool boiling. Increasing the heat flux lead to a limiting state of flow where the splash of droplets from the heated surface was no longer increased. It was found that a close relation existed between the critical heat flux and the jet velocity over a wide experimental range of jet velocities ranging from 1 m/s to 60 m/s. This relation is expressed in Equation 2.1. It was suggested by the authors that the critical heat flux may be connected to the separation of the liquid flow from the heated surface accompanied with the effusion of vapor.

\[
q_{CHV} = 3.4 \cdot 10^6 U^{0.39}
\]  

(2.1)

Ma and Bergles [27] attempted to show that jet impingement cooling is a possible means of accommodating the high heat fluxes which result from testing microelectronic chips at power levels well above those expected during normal operation. In this experimentation an apparatus was developed that utilized normally impinging circular
submerged jets of saturated or subcooled R-113. From their research, it was found that the boiling curve hysteresis with jet boiling is much less than with free convection. This is a definite plus for jet cooling, since the large hysteresis measured in many free convection environments is not acceptable in many microelectronic chip applications. It was also found that for a given experiment, the boiling curves for various jet velocities merge at higher heat fluxes to define a single established flow boiling curve.

Ma and Bergles also recorded that by impinging the surface with a subcooled jet of R-113, critical heat flux occurs at a heat flux on the magnitude of $10^6$ W/m² when the subcooling is approximately 30 K. Critical heat fluxes were found to vary as the cube root of the jet velocity, but are only weakly dependent on subcooling at higher values of subcooling. It should be noted that the critical heat flux data recorded in said experimentation was higher than data reported by previous investigators for uncertain reasons.

Conflicting data to that collected by Ma and Bergles was recorded in a series of experiments conducted by Monde and Katto [28]. In their research, they found that the critical heat flux was proportional to the square of the subcooling. The reason for the conflicting results may be due to the fact that the investigations conducted by Monde and Katto only considered subcooling up to 16 K. Another possible reason for the discrepancy could be that the experiments conducted by Ma and Bergles were conducted only under submerged conditions while Monde and Katto’s data were for a free jet. The generalized correlation of the critical heat flux data for the boiling system utilized in this experimentation, including the impinging jet, was determined to be as shown in Equations 2.2a and 2.2b:
\[ \frac{q_{\text{cor}} / \rho_s h_{\text{v}}}{U} = 7.45 \cdot 10^{-3} \left( \frac{\rho_s}{\rho_v} \right)^{0.725} \left( \frac{\sigma}{\rho_v U^2 D} \right)^{1.3} \cdot (1 + \varepsilon_{\text{cor}}) \]  

(2.2a)

where \( \varepsilon_{\text{cor}} \), the correction factor for liquid subcooling, is given as:

\[ \varepsilon_{\text{cor}} = 2.7 \left( \frac{\rho_v}{\rho_s} \right)^{0.5} \left( \frac{c_v (T_{\text{sat}} - T_{\text{v}})}{h_{\text{v}}} \right)^{3.0} \]  

(2.2b)

### 2.3 Disadvantages of Impingement Jet for Heat Dissipation

Even though it has been proven that impingement by a turbulent jet greatly increases the amount of heat transfer that is possible from a flat heated surface, there are drawbacks inherent within this technology. Experiments by Gu et al. [29] with a synthesized hydrocarbon fluid revealed that jet cooling supports the highest local heat transfer coefficient, however only in the stagnation region. Outside of this zone, the heat transfer diminishes rapidly resulting in large surface temperature gradients. The reason for this rapidly decreasing heat transfer is supported by an analysis conducted by Saad and Antonides [30]. According to their analysis, in the case of a circular jet impinging on a solid plate, the fluid pressure (gauge) at the surface of the plate falls to about 20% of the stagnation pressure at a distance of one and a half times the jet radius, and then to about 2.5% at a distance of twice the jet radius, where the dynamic pressure becomes
negligibly small. This decreasing dynamic pressure results in an increasing temperature profile from that measured at the center of the heated surface.

Some of the additional disadvantages of jet impingement cooling have been addressed by Zumbrunnen and Aziz [31]. Their single-phase experimental investigation with water focused on the effects of flow intermittency on the heat transfer to a impinging jet. Enhanced heat transfer was achieved by periodically restarting the flow and thereby forcing renewal of the boundary layer.

More recent work by Estes and Mudawar with FC-72 and FC-87 [32] revealed some additional disadvantages of jet cooling. For their investigation, the jet separated from the surface during vigorous boiling due to the momentum of the vapor normal to the surface. They showed how spraying drops onto the heated surface were more effective at securing liquid film contact with the surface.

Comparisons of heat transfer have been made between heat transfer rates that can be achieved with jet impingement and spray cooling. Cho and Wu [33] found that jet impingement cooling with Freon 113 created a large dry area on the surface when the critical heat flux was approached. However, for spray cooling, a liquid film with nucleated boiling was maintained for the duration of the experiment. At a given liquid flow rate, spray cooling was shown to produce a higher critical heat flux. From their research, they concluded that the spray velocity and flow rates were the controlling parameters for the critical heat flux and a modified jet impingement correlation for the critical heat flux was suggested that is based on the Weber number, We.

\[
\dot{q}_{\text{crit}} = 1.345(\text{We})^{0.319}
\]

(2.3)
The major benefit of immersion cooling when compared to solid-solid interface conduction is the elimination of the thermal resistance that is inherent within this type of heat transfer. There are two distinct types of immersion coolers: single-phase direct liquid cooling and phase-change direct liquid cooling. Natural convection, single-phase, immersion cooling is of interest in thermal management of components dissipating less than approximately 1W/cm² [4]. These components may be encountered both individually and in arrays of heated elements.

Natural convection, in vertical, parallel arrays of printed circuit boards, was examined in the analytical and experimental study by Bar-Cohen and Schweitzer [34]. Using flat plate correlations, to approximate heat transfer from densely packaged printed circuit boards, the authors derived and empirically confirmed composite Nusselt Number relations for varying channel widths in both symmetric and asymmetric heating configurations. The Bar-Cohen and Schweitzer relation for the symmetrically-heated isoflux channel Nusselt Number, based on the midheight temperature is shown in Equation 2.4.

\[
Nu_{h,\text{sym}} = \left[ 2 \left( \frac{Ra}{Re} + 1.8 \left( \frac{Ra}{Re} \right)^{2/3} \right) \right]^{1/3}
\]  

(2.4)

Of the various phase change processes, the nucleate boiling regime is the most efficient, with surface superheat only weakly dependent on heat flux and heat transfer coefficients that equal approximately 1 W/cm²K. It is for this reason that nucleate boiling is of primary interest for thermal control of high heat flux devices. The nucleate
boiling regime lies between boiling incipience, associated with a steady release of vapor bubbles from distinct nucleation sites, and the peak nucleate, or critical, heat flux, associated with vapor blanketing of the heated surface [4].

The critical heat flux is the value at which the transition from nucleate boiling to film boiling occurs and, for all but cryogenic liquids, represents the maximum allowable component dissipation. The Zuber [35] correlation, shown as Equation 2.5, relates vapor blanketing of the heated surface, in pool boiling, to the Helmholtz hydodynamic instability of vapor columns emanating from discrete nucleation sites.

\[
q_{\text{cr}} = \frac{\pi}{24} \rho_v^{1/3} h_f \left[ \sigma \left( \rho_v - \rho_l \right) \right]^{1/3}
\]

(2.5)

This relation is commonly used to determine the transition from nucleate to film boiling (critical heat flux). Extensive research into this phenomena during the past 30 years has revealed that Equation 2.5 is best used to represent the critical heat flux on a large, thick-walled, horizontal surface in contact with a highly-wetting, saturated liquid [36]. Relatively small heaters and boiling in subcooled liquid lead to substantially larger CHF values [37], while thin-walled heaters and boiling of non-wetting liquids produce significantly lower CHF values than given in Equation 2.5 [38].

2.5 Streaming Forced Convection

When natural convection is insufficient to maintain component temperatures in the desired range, the heat transfer requirements can often be met by use of single-phase,
streaming forced convection heat transfer (i.e., flow parallel to the axial dimension of the component). High velocity flow of fluorocarbon liquids, along millimeter-size heaters can result in heat transfer coefficients as high as the 3.3 W/cm²K, achieved by Samant and Simon [39]. Single phase flows may be laminar, transitional, or turbulent with mixed convection effects likely to be significant in the low Reynolds number and high Rayleigh number flow encountered in direct cooling with fluorocarbon liquids [40].

At the onset of nucleate boiling, a flow past the heated element, at velocities faster than the natural bubble departure velocity, can significantly increase the amount of heat transfer from the surface. If a very high velocity fluid is blown past a small heated surface, the total elimination of the nucleate boiling regime is possible [41]. It has also been shown that for subcooled flow boiling on heaters embedded in the walls of a channel that for a given fluid, pressure level, and surface conditions, the relationship between superheat and heat flux in fully-developed nucleate boiling has been found to be independent of velocity of crossflow and subcooling [42].

Critical heat flux data and correlations, for chip-like heaters embedded in high velocity channel walls, have been reported by several recent investigators. Lee and Simon [43] correlated the results of an extensive FC-72 boiling study, involving nichrome and platinum heaters, with heating lengths between 0.25 mm and 3 mm, liquid velocities between 1 m/s and 17 m/s, and liquid subcoolings ranging from 13.2 K to 68 K, as shown in Equation 2.6. A peak critical heat flux of 426 W/cm² was observed in this study.

$$\frac{q_{crit}}{\rho_f h_g U} = 0.04 \left( \frac{\rho_i}{\rho_f} \right)^{0.89} \left( \frac{\sigma}{\rho_f U^2 L} \right)^{0.33} \left[ 1 + 3.03 \left( \frac{\rho_i}{\rho_f} \right)^{0.37} \left( \frac{c_p \Delta T_{sub}}{h_g} \right)^{0.42} \right] \quad (2.6)$$

21
Mudawar and Maddox [44] reported FC-72 critical heat flux data for a rectangular channel with a single 12.7 mm heater, covering a wide range of velocities and subcoolings, and yielded peak heat fluxes in excess of 100 W/cm². From the results of their analysis, two distinct velocity regions with two entirely different \( \frac{d\eta_{\text{eff}}}{dU} \) slopes. The change in slope was found to be related to a visually observed change in flow pattern from vapor in the downstream liquid to vapor clots that randomly cover the surface.

Within the last 20 years, much research has been conducted in order to increase the amount of heat dissipated from a heated surface in a crossflow by impinging the heated surface with submerged jets. Cole and Hollworth [45] performed tests where staggered arrays of round turbulent air jets impinged upon a flat heated surface. The spent air was constrained in such a way as to exit at one end of the test section, thus establishing a crossflow. For this experiment, the airflow was varied to achieve a range of mean jet Reynolds number from 2500 to 25000. A typical result from the author’s experimentation was a wall heat flux of 15 kW/m² for a jet Reynolds number of 4900.

From this research, it was also observed that heat transfer peaks may be shifted considerable distances downstream by the induced crossflow; deflections of as much as four jet diameters were observed in these tests. Because of this, the authors recommend that it would be good design practice to extend the cooled surface somewhat past the end of the jet array as to take advantage of the “residual” cooling from the (deflected) last row of holes. It should also be noted that at constant coolant flow per unit heat transfer area, arrays of jets with low hole density consistently yield higher average heat transfer than arrays with more holes per unit area.
CHAPTER 3

EXPERIMENTAL APPARATUS AND PROCEDURES

For this research, a cavitation jet was used to dissipate heat from a heated surface for two different boiling scenarios: pool boiling in stagnant water, and pool boiling in a channel flow environment. To test the jet’s efficacy in dissipating heat in the first scenario, a water container was developed such that a heater could be mounted in its bottom and the jet could be positioned directly above its center. This container also had the capability of being sealed at its top such that air could be removed from the system. In contrast, the second set of experimentation was conducted in a channel flow cell in which water was injected in one side and evacuated from another. This cell was constructed such that a heater could be mounted in its bottom and the jet could be positioned directly above its center. Both of these apparatuses were constructed from materials such that photographs of the bubbles being removed from the heaters could be easily photographed from their respective exteriors.

3.1 Physical Properties of Piezoelectric Diaphragm

3.1.1 Input Voltage, Holder Diameter, and Bath Temperature

Of the many factors that can affect the resonant frequency of an oscillating surface, the inner diameter of the surface onto which it is mounted as well as the temperature of the bath water into which it is submerged are among the most critical. The latter is especially true when the material that comprises the diaphragm is soft, such
as brass. The piezoelectric driver chosen for this experimentation is comprised of a 0.39-mm-thick circular brass disk having a diameter of 18.2 mm driven by a concentrically mounted 0.19-mm-thick piezoceramic wafer having a diameter of 11.6 mm. This particular diaphragm type used in this experimentation was manufactured by Murata and was part number 7BB-18R2-G2602. This particular driver was chosen because of its ability to achieve the surface velocity necessary for the creation of the cavitation jet.

Three experiments were conducted in order to quantify the affect that each of the following had on the performance of the aforementioned piezoelectric diaphragm: the input voltage to the driver, the holder diameter, and the bath temperature. These tests were each begun by gluing a piezoelectric driver to the end of an 8 cm long delrin tube as shown in Figure 3.1. This tube was positioned such that the end of the tube to which the diaphragm was glued was submerged beneath 3 cm of distilled water in a 100 ml beaker. The glue used to mount the brass disk on the delerin rod was a 100:8.5 weight mixture of

![Figure 3.1: Experimental set-up used to quantify properties of piezoelectric diaphragm.](image)

24
Stycast 2651 and Catalyst 11, both from Emerson & Cuming. The assembly was allowed to cure for one hour at 120°C. This glue selection was another crucial element in creating the cavitation jet from the oscillating diaphragm. This glue was not only waterproof; it was extremely resilient against fluctuations in modes of vibration and temperature change. In both experiments, the surface velocity was monitored by a Polytec OFV3001 laser vibrometer positioned such that its laser was fixed onto a mirror, which directed it onto the ceramic surface of the driver. The output of the vibrometer was in \( V_{\text{rms}} \), so a Fluke 45 Dual Display Multimeter was required for its display. The vibrometer setup has an inherent uncertainty of approximately 6 mV.

3.1.1.1 Procedure for determining diameter dependency

To test the dependence of the diaphragms' resonant frequency on the inner diameter of its holder, three identical diaphragms were glued to three delrin rods, having identical outer diameters and inner diameters of 17 mm, 15 mm, and 13 mm. To begin each test, a frequency scan was conducted to audibly determine the location of the resonance frequency for each of the three driver assemblies. Each actuator was driven at the resonance frequency of its first axisymmetric mode of vibration using a laboratory function generator and a high voltage \( (\max. 120 \ V_{\text{rms}}) \) broadband (25 Hz-150 kHz) amplifier. Once the approximate location of the resonant frequency of each assembly was noted, each assembly was placed separately into a beaker and the laser from the vibrometer was positioned onto the ceramic portion of the driver as shown in Figure 3.1. For each assembly, a frequency scan was conducted beginning at 2 kHz less than the predicted resonance frequency and ending 2 kHz greater than the predicted resonance.
frequency. These scans were conducted in 0.1 kHz increments, and for each frequency, the $V_{rms}$ output of the vibrometer was recorded.

3.1.1.2 Procedure for determining temperature dependency

To test the dependence of the diaphragms' resonant frequency on the temperature of the bath water into which it is submerged, a single piezoelectric diaphragm was glued to the end of a delrin rod and placed within a beaker as shown in Figure 3.1. The beaker was then placed on top of a hot-plate such that the water within the bath could be heated to boiling. Two $T$-type thermistor thermocouples were positioned on opposite sides of the driver assembly such that the temperature of the water in the vicinity of the driver could be monitored. As was done in the diameter experimentation, the laser from the vibrometer was fixed onto the ceramic portion of the driver such that its surface velocity could be obtained. This experiment was conducted by gradually increasing the temperature of the bath water such that the temperature of the diaphragm could be assumed to be identical to that of the water. For eight different temperature levels, ranging from 20°C to 90°C, the $V_{rms}$ output of the vibrometer was recorded.

3.1.1.3 Procedure for determining voltage dependency

In order to determine what affect that the input voltage supplied to the piezoelectric diaphragm has, a single diaphragm was glued to the end of a delrin rod, with an inner diameter of 17 mm, and placed within a beaker as shown in Figure 3.1. For this experiment, the laser from the vibrometer was fixed onto the ceramic portion of the driver such that its surface velocity could be obtained. After finding the resonant
frequency of the driver at the nominal input voltage of 90 \( V_{\text{rms}} \), the voltage was gradually lowered in 10 V increments while keeping the frequency set at the predetermined resonant frequency. At each of these voltage levels, the output of the vibrometer was recorded so that the velocity of the surface, at each input voltage level, could be calculated.

3.1.2 Threshold Surface Velocity Determination

The purpose of this test was to determine the threshold surface velocity of the piezoelectric diaphragm necessary for jet formation. This test was begun by gluing a piezoelectric driver to the end of an 8 cm long delrin tube as shown in Figure 3.1. This tube was positioned such that the end of the tube to which the diaphragms was glued was submerged beneath 3 cm of distilled water in a 100 ml beaker. The laser vibrometer was positioned such that its laser was fixed onto a mirror, which directed it onto the ceramic surface of the driver. The output of the vibrometer was in \( V_{\text{rms}} \) so a Fluke 45 Dual Display Multimeter was required for its display. Data was recorded over a 5 kHz span in 0.2 kHz intervals, and the point where the jet began and ended was diagnosed by observation of the wet surface of the driver.

From the voltage output of the vibrometer, Equation 3.1 was used to calculate peak-to-peak velocity of the oscillating surface with an uncertainty of \( \pm 2.12 \) mm/s.

\[
Vel_{pp} = \left( V_{\text{rms}}, \ 2\sqrt{2} \right)x
\]  

(3.1)
From this velocity, the peak-to-peak surface displacement, $\delta$, was calculated using Equation 3.2.

$$\delta = \frac{V_0}{2\pi f}$$  \hspace{1cm} (3.2)

To obtain the displacement from the resting position of the diaphragm, the result obtained from Equation 3.2 was divided by two. From this displacement, the acceleration, $A$, of the surface of the diaphragm was calculated using Equation 3.3 at an uncertainty of ± 60 m/s^2.

$$A = \frac{\delta}{2(2\pi f)^2}$$  \hspace{1cm} (3.3)

### 3.2 Pool Boiling Test Apparatus

#### 3.2.1 Actuator Assembly

The actuators used for this experimentation were identical to those used

![Figure 3.2: a) solid model of actuator assembly, b) cross-section of piezoelectric diaphragm holder.](image)
in the property analysis experimentation. In mounting the actuator, the brass disk was glued concentrically over a hollow chamber with a diameter of 15 mm and a 13 mm depth contained within an acrylic cube measuring 25 mm X 25 mm X 19 mm. The cube was fitted with a 3.2 mm X 10-32 nylon elbow such that the mounting chamber could be provided with air via an air tube. This elbow could be rotated as to provide air to the dry side of the driver, regardless of the position of the assembly. The piezoelectric diaphragm and assembly are depicted in Figure 3.2.

In order to adjust the distance between the diaphragm and the heated surface as well as the angle at which the jet is impinging the heated surface, the acrylic cube was bolted to two aluminum shims each measuring 30.5 mm X 6.3 mm X 25.4 mm. Each of these shims contained six height adjustment holes at 3.8 mm increments along their respective vertical centerlines. Figure 3.3 depicts these shims fastened to the actuator assembly described in Figure 3.2.

3.2.2 Test Module Design

The test module was designed such that the actuator assembly could be placed

![Diagram](image)

**Figure 3.3:** a) Solid model of shims bolted to actuator assembly, b) part drawing of shims used to adjust the height and angle of driver.

29
directly over an Intel thermal test die to allow for direct impingement by the cavitation jet. Each piece of the module assembly was machined from clear acrylic to allow for ease of viewing and photography. Figure 3.4 depicts the circular wall of the test module.

The top of the test module was designed to provide an airtight enclosure for the actuator assembly for reduced pressure testing. To do this, a groove was ground into the top such that it could fit snugly over the module wall. Inside this groove, an additional groove was ground to provide room for an O-ring. The purpose of this O-ring was to not only provide an airtight environment but also to keep the testing fluid from leaking from the module. To ensure that both of these tasks were accomplished, the O-ring was coated with a non-curing silicone provided by Dow Corning. The O-ring selected for use in the module top was made by Buna-N and had an AS568A dash number of 160. In addition to the groove, three holes were tapped into the top such that they would provide access to the central area of the module when assembled. The first of these holes, measuring 1/2-10, was used to remove air from the test module via a Swagelock fitting and an air tube.

![Figure 3.4: a) solid model of module wall, b) CAD drawing of module wall.](image)
The second of the three holes, measuring 1/4-20, was used to insert a pressure transducer into the module. The last hole was used to gain access to the wires running from the piezoelectric diaphragm. Once the appropriate tools had been inserted into each of the three holes, each was sealed using a non-flow RTV. Outside of the central area of the module top, four additional holes were drilled such that guideposts could be inserted to align the module assembly. Figure 3.5a depicts the module top and Figure 3.6 shows the detailed part drawing associated with its design.

The module base was designed and built much like the module top. The difference between these two parts is the portion of each that falls within the central chamber of the module. There were no holes drilled into the module base, but instead, a square chamber was created such that the Intel thermal test vehicle could be mounted with its heated surface exposed to the fluid within the module. The test die was glued to the module base using a non-flow RTV. Figure 3.5b depicts the module bottom and Figure 3.7 shows the detailed part drawing associated with its design.

When fully assembled, the test module test apparatus appeared as depicted in Figure 3.8.

**Figure 3.5:** a) solid model of module top, b) solid model of module base.
Figure 3.6: Part drawing of test module top.

Figure 3.7: Part drawing of test module bottom.
Figure 3.8: Pool boiling test assembly.

Each of the pool boiling experiments began by filling the small circular water tank depicted in Figure 3.8 with distilled water such that the actuator assembly was submerged while leaving the air tube exposed. In the bottom of the tank, an Intel Pnetop thermal test die was mounted such that its heated surface was exposed to the interior of the water tank while its pins were exposed beneath the tank. In order to take more precise measurements, the heat spreader cap was removed from the chip thus exposing the 1.18 cm$^2$ silicon wafer beneath. The output of the imbedded diodes within the test die was sent to a Fluke NetDAQ data acquisition system and was displayed and recorded using software provided by Fluke. The power supplied to the test die for heating was provided by a 900 W Sorensen DCR 150-6B power supply. The actuator was driven at the resonance frequency of its first axisymmetric mode of vibration (nominally 7 kHz) using a laboratory function generator and a high voltage (max. 120 V$_{ rms }$) broadband (25
Hz-150 kHz) amplifier. A schematic of this experimental set-up is depicted in Figure 3.9.

![Schematic of pool boiling experimental apparatus.](image)

**Figure 3.9: Schematic of pool boiling experimental apparatus.**

### 3.3 Pool Boiling Test Procedures

#### 3.3.1 Optimum Height Determination

The purpose of this test was to determine the optimum separation distance between the surface of the diaphragm and the heated surface. This optimum distance is the point at which the overall heat dissipation from the heated surface is at its highest when compared to boiling heat transfer with no jet. As mentioned before, each of the shims had six equally spaced holes with the first one being at 7.6 mm and the last being at 26.7 mm. To set the separation distance between the diaphragm and the test die, the acrylic block was bolted to the desired holes in each of the shims, and the assembly was placed in the test module above the Intel thermal test die as shown in Figure 3.8. After driving the diaphragm to its first mode of vibration at a voltage of approximately 90 Vrms, power was sent to the test die in 5 V increments. For each increment, the temperature of
the test die was allowed to come to steady state and then was recorded. This process was repeated until the temperature of the test die was within 5°C of its maximum capacity, 120°C. Similar sets of data were recorded for each of the six height adjustments.

3.3.2 Optimum Angle Determination

The purpose of this test was to determine the optimum angle of impingement between the surface of the diaphragm and the heated surface. This optimum angle is the point at which the overall heat dissipation from the heated surface is at its highest when compared to boiling heat transfer with no jet. After determining the optimum separation distance between the diaphragm and the test die, the acrylic block was set to this height by bolting it to the appropriate holes on the shims. The angle of the diaphragm was established by rotating the acrylic block to the appropriate angle using the bolts as the point of rotation. For each angle, the placement of the center of the diaphragm with respect to the center of the test die was altered as to maintain a constant distance between the two surfaces. After placing the actuator assembly back into the water tank, the diaphragm was driven to its first mode of vibration at a voltage of approximately 90 V_{rms}. Following this, power was sent to the test die in 5 V increments. For each increment, the temperature of the test die was allowed to come to steady state and then was recorded. This process was repeated until the temperature of the test die was within 5°C of its maximum capacity, 120°C. Similar sets of data were recorded for four different impingement angles. Figure 3.10 depicts the various parameters measured in both the impingement angle and separation distance optimization tests.
3.3.3 Efficacy of Turbulent Jet for Heat Removal

This experiment was designed to quantify the amount of additional heat removal that was made possible by impinging the thermal test die with the turbulent jet oriented at its optimal distance and angle. In this analysis, temperature data was obtained from the thermal test die for two different scenarios: pool boiling without the jet actuated and pool boiling with direct impingement from the turbulent jet. In each case, the test die was sent power in 5 V increments, and for each increment, its surface temperature was allowed to reach steady state and then was recorded. This process was repeated until the temperature of the test die was within 5°C of its maximum capacity, 120°C. For the case where the jet was being actuated, the diaphragm was driven to its first mode of vibration at a voltage of approximately 90 Vrms. The recorded power supplied to the chip has an uncertainty of ±0.035 W, and the uncertainty associated with the temperature measurements obtained from the diodes planted within the chip is ±0.71°C.
3.3.4 Reduced Pressure Analysis

The motivation of the reduced pressure analysis was to lower the saturation temperature of the distilled water within the test module such that two-phase heat transfer could begin at a temperature less than 100°C. To do this, the air was evacuated from the test module using a Welch vacuum pump Model No. 2022B-01. The pressure within the test module was monitored using an Endevco pressure transducer Model No. 8541-50 and Serial No. 10008 with a sensitivity of 5.57 mV/psi. Since the output of the pressure transducer was in mV, it was displayed using a Fluke 45 Dual Display Multimeter. The pressure within the chamber was calculated by dividing the output voltage by the sensitivity of the pressure transducer.

In order to observe the interaction between the cavitation jet and the water into which it was submerged, it was important to remove as much of the dissolved air from the water as possible. Since the reduced pressure analysis was to simply observe the jet in a low pressure environment, no precise method of air removal was implemented. To begin removing the air from the bath, the water was heated above its saturation temperature such that boiling began. The water was allowed to boil for approximately 30 minutes before being removed from the heat and sealed as to not allow additional air into the test chamber. After sealing the heated water in the test chamber, air was forcibly removed via a vacuum pump. Once the pressure was reduced past approximately 0.023 bar, the test chamber was shaken as to dislodge the newly developed bubbles from the chamber’s walls. After this step, the pressure in the chamber was allowed to gradually increase until it was in equilibrium with ambient. This process of evacuating the air from the chamber, shaking the bubbles loose, and allowing it to return to ambient pressure was

37
repeated until no additional bubbles formed after applying a vacuum to the system. Once this stage was reached, the water was deemed deaerated enough to adequately witness the influence of the jet at the reduced pressure levels.

For visualization of the flow within the cell, the water inside the test module was seeded with DF-L-520AR dedusted aluminum flakes at a concentration of approximately 2,000 particles/ml. Before introducing these flakes to the water within the test module, the flakes were treated with a mixture of detergent and water. To do this, the flakes, detergent and water were poured into a beaker in a 1:1:5 ratio. After stirring the mixture for at least 1 minute, the mixture was allowed to settle for approximately 30 minutes. After settling, the flakes that remained aloft within the water were poured into the test module until the desired concentration was achieved.

To obtain a uniplanar view of the water jet, a Kodak Motion Corder SR-Ultra high-speed camera was angled perpendicularly to a 1 cm deep light sheet cast through the water by a 250 W fiberoptic light source shining through a 38 cm diameter acrylic cylinder. The light source, coupled with the neutrally buoyant flakes, provided a clear visualization of the flow pattern of the cavitation jet at the different pressure levels.

3.4 Channel Boiling Test Apparatus

3.4.1 Test Cell Design

The motivation of the channel boiling test cell was to provide cool bulk fluid to the cavitation jet in an effort to provide additional heat transfer from the heated surface onto which the jet is impinging. The test cell was machined from a solid block of acrylic measuring 25.4 mm X 50.8 mm X 77.4 mm. Two of the four 25.4 mm tall sides of the
cell were retrofitted with windows as to allow for easy viewing of the activities commencing within the cell's internal chamber. Into the two remaining sides were drilled two 1/4 NPT tapped holes such that SwageLock fittings could be screwed into opposing sides of the cell. To provide an opening in which to insert the piezoelectric diaphragm into the cell, a 19 mm hole was drilled into the top of the cell. As was the case with the test module, the bottom of the cell was machined such that an Intel thermal test die could be inserted with its heated side exposed to the fluid inside the cell. Figure 3.11 is a solid model of the test cell, while Figure 3.12 is an assembly of the various part drawings associated with the construction of the test cell.

Two series of experiments were conducted using the channel flow test cell. In the first set of experimentation, the test cell was glued to the surface of an Intel chipboard such that the test die was positioned with its heated surface exposed to the fluid within the cell. In the other set of experimentation, a calibrated copper heater was utilized as the test surface because of its ability to reach much higher temperatures than the test die. Both heaters were sealed into place using a non-flow RTV silicon sealant.

Figure 3.11: Solid model of channel boiling test cell.
Figure 3.12a: Part drawing of side of cavitation heat transfer cell.

Figure 3.12b: Part drawing of bottom of cavitation test cell.

For all of the experimentation conducted using the flow test cell, the cavitation jet was created in the same manner as was used in the pool boiling experimentation. In order to place the jet within the flow cell, the piezoelectric driver was glued to the end of a 7.6 cm long hollow brass tube having an outer diameter of 19 mm and an inner diameter of 16 mm. The driver was glued to the end of the brass tube with a 100:8.5 weight mixture of Stycast 2651 and Catalyst 11 cured for one hour at 120°C. The driver
and brass tube assembly was inserted into the 19 mm hole located in the top of the test cell until the surface of the driver was located at the optimum distance as was quantified in the pool boiling experimentation. Once the brass tube was securely in place, a thin ribbon of flow RTV was applied around the circumference of the brass tube at the point where it met the test cell. Figure 3.13 is a photograph of the test cell glued to the surface of the Intel chip board.

Each of the channel boiling experiments began by switching on the VWR Scientific Model 1156 control temperature water bath connected in series with the test cell. Since the test cell is sealed, air would become trapped above the waterline as the water filled the test cell. To evacuate this air from the cell, the apparatus was tilted such that the air was allowed to leave through its exit. The flowrate of the water flowing through the cell was controlled with a King 7-gph flow controller located at the exit of the test cell. For the experiments that included the use of the Intel test die as a heat source, the chip preparation is exactly the same as what was used in the pool boiling.

![Figure 3.13: Photograph of channel flow heat transfer cell.](image-url)
experimentation. In order to take more precise measurements, the heat spreader cap was removed from the chip thus exposing the 1.18 cm$^2$ silicon wafer beneath. The output of the imbedded diodes within the test die was sent to a Fluke NetDAQ data acquisition system and was displayed and recorded using software provided by Fluke. The power supplied to the test die for heating was provided by a 900 W Sorensen DCR 150-6B power supply. The actuator was driven at the resonance frequency of its first axisymmetric mode of vibration (nominally 7 kHz) using a laboratory function generator and a high voltage (max. 120 V$_{max}$) broadband (25 kHz-150 kHz) amplifier. A schematic of this experimental set-up is depicted in Figure 3.14.

![Diagram of channel boiling experimental apparatus](image)

Figure 3.14: Schematic of channel boiling experimental apparatus.

42
3.5 High Heat Flux Calibrated Heater

Due to the high amounts of heat that can be dissipated by the cavitation jet, a more robust heater was used to study cavitation jet impingement cooling at higher heat fluxes. The heater used in this experimentation was the same one used in the experimentation conducted by Sellers [1], with only a few modifications. Sellers based her heater design on the one created by Sheffield [46], since it had proved very reliable at heat fluxes up to 400 W/cm². However, several changes were necessary, including a smaller mass for the copper block as well as a square test surface. The new design incorporated a 8.0 mm by 8.0 mm test surface, and it contained only about one-fifth the copper used in Sheffield’s original design. The following text was taken directly from the thesis written by Sellers [1] with the proper modifications made to reflect the necessary changes in her design.

3.5.1 Heater Design

The main component of the heater assembly, shown in Figure 3.15, was an internally heated block that was constructed from a 22.23 mm diameter copper bar stock. The cavitation jet impacted the exposed upper surface of the heater, while a single cartridge heater imbedded in the base of the copper provided heat to the heater block. The heater block was instrumented with T-type thermocouples to monitor the internal temperature as well as to measure temperature gradients at the upper surface. The temperature gradients were used to calculate surface temperature and heat transfer rate at the wetted surface.
Heat input to the copper block was provided by a Watlow FIREROD cartridge heater having a diameter of 9.53 mm. The cartridge heater provided up to 400 W at the input voltage of 120 V, and it was coated with Omegaetherm 201, a high thermally conductive paste, before it was inserted into the copper. The cartridge heater was held securely in place by a circular groove ground into the base of the heater assembly.

For the temperature-controlled investigation, the input to the cartridge heater was varied and provided by a 240 V Slide Regulator Variac. The controller, in a switched DC configuration, provided an open or closed circuit to a solid state relay, which provided up to 120 VAC to the cartridge heater (voltage limitation due to local power supply constraints).
As shown in Figure 3.16, the lower region of the copper cylinder was 22.23 mm in diameter by 34.93 mm in length and it contained the electrical heat source. A short length of 6.35 mm was used to transition the copper cross-section to a square section with dimensions of 10 mm x 10 mm. This section enhanced the one-dimensional aspect of the heat flow to the wetted surface. At the top of the heater, the cross-section was reduced to 8 mm x 8 mm. This reduction in cross-section further increased the heat flux as well as provided an increased surface area to seal the copper to the waterproof cover.

Figure 3.15 shows a cutaway of the heater assembly. Insulation was used to reduce the heat loss from the surfaces other than the test region. The copper rested on a 12.7 mm thick piece of G-10 Fiberglass. The entire length of the copper was insulated with approximately 40 mm of Carborundum Fiberfrax Blanket, Durablanket #8. The ceramic-fiber blanket was wrapped around the entire length of the copper and was held in place by a polyvinylchloride (PVC) pipe.

The top layer of insulation was covered by a G-10 Fiberglass shield. The shield had a 8.0 mm x 8.0 mm tapered hole that tightly fit around the test section of the copper. A seal of VersaChem high temperature silicone (type 650) was made between the copper and fiberglass to prevent any water from leaking into the test apparatus. Any water that infiltrated the enclosure had the potential of increasing the two-dimensional heat transfer in the neck region of the copper.

The exposed tip of the copper heater provided the heat transfer surface for the impinging jet. The surface was initially polished with FEPA 2400 grit polishing paper. The surface was then cleaned multiple times with a polishing agent, BRASSO. Following the polishing, the surface was rinsed with solvents in the following order:
acetone, ethanol, and isopropyl alcohol. Several flushes with distilled water completed the cleaning routine. All cleaning occurred at surface temperature less than 40°C.

Figure 3.16: Detail of copper heater.
Figure 3.17: Detail of measurement region of heater.
3.5.2 Instrumentation System

The copper heater was instrumented with a total of nine thermocouples to monitor temperatures in the neck region of the heater. The ANSI (American National Standards Institute) T-type thermocouples were constructed of 30 gauge (diameter 0.2546 mm) copper and constantan wire. To electrically insulate the thermocouple bead, the bare junction was coated with a film of Omega Thermcoat SL silicone phenolic lacquer and cured for one hour at 150°C. The thermocouple assemblies were snugly fit into holes filled with Omegatherm 201, a high thermally conductive paste. The ceramic insulators prevented contact between the two dissimilar metals except at the junction as well as properly centering the thermocouple bead within the hole. Figure 3.16 shows the placement of the nine thermocouples within the upper portion of the heater. Three thermocouples were placed at each of three levels. At each level, the thermocouples were embedded to a depth of 2.54 mm within the copper. In order to obtain the temperature measurements from the voltage outputs of the embedded thermocouples, all of the thermocouples were wired into a National Instruments SXCI 1303 terminal block which was read by a National Instruments SXCI 1000 chassis. Each measurement was displayed and logged by a LabVIEW data acquisition system.

3.5.3 Surface Temperature and Heat Flux Measurements

Figure 3.17 shows the neck of the heater and the placement of the three levels of thermocouples used to measure the temperatures for the heat flux calculations. At each level, four thermocouples were used to compute an average temperature. The two thermocouples levels closest to the tip, \( T_1 \) and \( T_2 \), were used to compute the temperature
gradient in the neck of the heater. Using Fourier’s Law, the heat flux could be calculated from the following expression,

\[ q_n = \frac{A_T}{A_i} \frac{k}{\Delta z} (T_i - T_f) \]  

(3.4)

where the cross sectional area of the heater at the thermocouple location, \( A_T \), and at the surface, \( A_i \), are known. The heat flux has an uncertainty of approximately 8.6%. The thermal conductivity, \( k \), for the copper was taken as 392.2 W/mK and the distance between thermocouple levels, \( \Delta z \), as well as between the surface and first level of thermocouples, \( \Delta z_{i1} \), was 3.18 mm. The temperature gradient between location 1 and 2 was used to extrapolate a value for the surface temperature, \( T_s \),

\[ T_s = T_f - \left( T_i - T_f \right) \left( \frac{\Delta z_{i1} - \Delta z_{i2}}{\Delta z} + \frac{A_{TC}}{A_i} \frac{\Delta z_{i2}}{\Delta z} \right) \]  

(3.5)

where \( T_f \) and \( T_i \) are the average temperatures at axial locations 1 and 2 respectively and \( \Delta z \) is shown in Figure 3.17. This temperature has an uncertainty of approximately 1%.

### 3.6 Channel Boiling Test Procedures

#### 3.6.1 Thermal Test Die Heat Source Experimentation

This procedure is for each of the experiments conducted using the thermal test die as a heat source is as follows. After mounting the cavitation jet heat transfer module to the board onto which the test die was fixed, the hoses were connected to the cell to allow for water from the water bath to flow through the cell. Once the cell was filled with
water and all air bubbles were removed from the cell, the driver was energized. After the jet formed, the thermal test die was powered on. The power dissipated by the die was measured by monitoring the steady-state voltage and current levels applied to the die. The heat flux removed from the die surface was estimated by dividing the total power dissipation by the surface area of the die (1.18 cm²). The temperature of the chip's surface was measured using the temperature diodes embedded within the chip and have an uncertainty of ±1°C. In each test, the test die was powered up in 5 V increments, and for each increment, the surface temperature was allowed to reach steady state before it was recorded. The test was terminated when the temperature of the test die was 115°C, 5°C lower than the maximum allowable value of 120°C. For cases in which the jet was on, the diaphragm was driven in its first mode of vibration with approximately 90 Vms.

3.6.1.1 Procedure for quantifying efficacy of cell for heat removal

In the first experiment, the water bath was set such that the water it provided to the heat transfer cell remained at a constant 20°C, and the flowrate of the water bath was set such that the water flowed at a rate of 5 ml/s through the cell. Data was collected for two different scenarios: channel boiling with the jet on and with the jet off.

3.6.1.2 Procedure for determining effect of increasing bath temperature

In the second experiment, the water bath was again set such that the velocity of the water traveling through the cell was approximately 5 ml/s, but in this experiment, the temperature of the water provided by the water bath was varied from 20°C to 60°C. At
both bath temperatures, data was collected for two different scenarios: channel boiling with the jet on and channel boiling with the jet off.

3.6.2 Calibrated Heater Heat Source Experimentation

The procedure for each of the experiments conducted using the calibrated heater as a heat source is as follows. After mounting the cavitation jet heat transfer module to the insulated surface through which the calibrated heater protrudes, water was allowed to flow through the cell and the actuator was energized. Using a Variac, the voltage to the heater was gradually increased as to not miss the onset of critical heat flux in the collected data. The heat flux dissipated by the cavitation jet module was measured by monitoring the temperature difference in the one-dimensional section of the heater. The heat flux removed from the heater is equal to the thermal conductivity multiplied by the temperature difference divided by the distance separating the thermocouples used to measure the temperature. By assuming the temperatures in the copper rod are linear along its length, the temperature of the heater's surface was extrapolated from the temperature data collected from the thermocouples. Each of the tests was terminated when the average temperature within the heater was approximately 200°C.

3.6.2.1 Procedure for determining influence of cell inclination

In the first experiment, the water bath was set such that the water it provided to the heat transfer cell remained at a constant 20°C, and the flowrate of the water bath was set such that the water flowed at a rate of 5 ml/s through the cell. The purpose of this test was to establish the cavitation jet's efficacy for increasing the amount of heat dissipated
from the surface of the calibrated heater at high surface temperatures. In addition to this, experimentation was conducted in an attempt to lengthen the amount of time that the jet would influence the heat transfer from the surface. To accomplish this, the cavitation jet heat transfer module was rotated 45° such that the water inlet was located below the water outlet. For both cases, data was collected for two different scenarios: channel boiling with the jet on and channel boiling with the jet off.

3.6.2.2 Procedure for determining flowrate influence

The purpose of this test was to characterize the affect that the crossflow rate has on the heat transfer enhancement capabilities of the cavitation jet. In this set of experimentation, the water bath was set such that the water it provided to the heat transfer cell remained at a constant 20°C and the flowrate of the water bath was varied from 1.5 ml/s to 7.5 ml/s in 2 ml/s increments.

3.6.2.3 Procedure for determining optimum separation distance

The purpose of this experimentation was to determine the effect that the separation distance between the surface of the calibrated heater and the surface of the driver has on the heat transfer capability of the cavitation jet. For this experimentation, the water bath was set such that the water it provided to the heat transfer cell remained at a constant 20°C, and the flowrate of the water bath was set such that the water flowed at a rate of 5 ml/s through the cell. To test the effect that separation has on the heat transfer, the separation distance was varied from 2.5 mm to 10 mm in 2.5 mm increments.

52
CHAPTER 4

RESULTS AND DISCUSSION

Each of the experiments included within this section was centered on the functionality of the vibration induced cavitation jet for enhancing heat transfer from a flat heated surface. Not only did this experimentation study the physical properties of the jet itself, but it attempted to quantify the capacity of the cavitation jet for improving heat transfer from a heated surface over that which is attainable using other heat transfer schemes.

Each experiment was centered on a two-phase cooling heat transfer process in which a submerged vibration-induced cavitation jet effectively dislodged small vapor bubbles attached to a solid surface and propelled them back into the cooler bulk liquid. This ejection technique involved forcibly removing the attached vapor bubbles with a submerged turbulent jet generated by a vibrating piezoelectric diaphragm operating at resonance. The piezoelectric driver induces pressure oscillations in the liquid near the surface of the driver, resulting in the time-periodic formation and collapse of cavitation bubbles that entrain surrounding liquid and generate a strong liquid jet. The resultant jet, which was directed at the heated surface, enhanced boiling heat transfer by removing attached vapor bubbles that insulated the surface while providing additional forced fluid convection on the surface. By coupling the cavitation jet with a crossflow within the test apparatus, the heat transfer was raised even further due to the fact that the flow swept the
bubbles downstream while keeping the temperature of the water within the cell regulated. Figure 4.1 is a flow visualization of the cavitation jet.

4.1 Physical Properties of Cavitation Jet

4.1.1 Threshold Surface Velocity for Jet Formation

The purpose of this experimentation was to quantify the threshold surface velocity that is necessary to produce a cavitation jet from the surface of an oscillating piezoelectric diaphragm. As seen in Figure 4.2, the minimum surface velocity required for jet formation was approximately 1100 mm/s ± 0.2 percent. This threshold velocity
Figure 4.2: Threshold surface velocity required for jet formation.

occurs at a surface displacement of approximately 0.012 mm and a surface acceleration of approximately 3300 g, at the nominal operating frequency of 9 kHz. The critical velocity, as shown in Figure 4.2, does not change with varying holder designs and bath temperatures, but the frequencies for which the jet forms are directly related to these parameters. This range does not signify the onset of cavitation, because cavitation occurs at lower velocities than what is required to produce the cavitation jet. It is not until this threshold is reached that the cavitation bubbles entrain the surrounding fluid and generate the cavitation jet flowing normal to the surface of the diaphragm.
4.1.2 Influence of Experimental Parameters on Surface Velocity

The formation of the cavitation jet is based on the velocity of the vibrating surface surpassing a certain threshold such that the jet forms from the cavitation that occurs at the interface between the fluid and the surface. Since this critical parameter is at its maximum at the resonant frequency of the driver, it is critical to be able to understand what influence various parameters have on the resonant frequency and thus the cavitation jet itself. There are many factors that play a role in determining at what frequency resonance occurs for a given diaphragm and holder assembly. These factors include: the inner diameter of the diaphragm holder, the temperature of the water bath, the depth that
the diaphragm is submerged beneath the surface of the bulk fluid, as well as the physical make-up of the diaphragm itself. From trial and error it was determined that the inner diameter of the diaphragm holder and the bath temperature are the most influential.

Figure 4.3 shows the influence that the bath temperature has on the surface velocity at the resonant frequency of the vibrating diaphragm. At a bath temperature of approximately 30°C, the surface velocity is approximately 1400 mm/s. At temperature above approximately 50°C, the diaphragm no longer has the surface velocity necessary for jet formation. The velocity decreases as the temperature of the bath water increases. This decrease approaches 300 mm/s for bath temperatures approaching the boiling point. This decrease in surface velocity is due to the changing dimensions of the vibrating diaphragm. Since the chosen diaphragm is brass (with a coefficient of thermal expansion of 19x10^-6 1/°C [47]) it is expanded considerably at the increased bath temperatures. The problem of thermal expansion was exacerbated by the fact that the diaphragm was rigidly adhered to its holder. This bond did not allow the axial expansion of the disk, so the disk was forced to expand in the direction of vibration thus dampening its vibration.

Figure 4.4 shows the influence of the inner diameter of the diaphragm holder with regards to the surface velocity of the vibrating diaphragm. Mounted to a holder with an inner diameter of 17 mm, the diaphragm achieved resonance at a frequency of approximately 6.3 kHz with a surface velocity of 1450 mm/s. By decreasing the hole diameter to 13 mm, the resonant frequency of the driver fell to approximately 11 kHz with a surface velocity of 1100 mm/s. Both the 17 mm and 15 mm hole sizes allow enough surface velocity to form a jet.
Figure 4.4: Effect of the inner diameter of the diaphragm holder on resonant frequency. Hole diameter = 17 mm (x), 15 mm (○), and 13 mm (▲).

The reason for both the decline in surface velocity as well as the increase in resonant frequency of the driver is directly linked to the ‘workable’ area of the diaphragm. As the inner diameter of the holder decreases, the area that is allowed to oscillate also decreases. Because of this decrease, the amount of flexion of the diaphragm’s surface decreases if the same input voltage is maintained. By decreasing the workable surface area of the diaphragm without changing its thickness, the frequency at which the diaphragm’s first axisymmetric mode of vibration occurs increases.

Figure 4.5 displays the influence that the input voltage to the driver has on the surface velocity of the driver operating at its resonant frequency. For input voltages less than 30 VAC, the amount of surface velocity is minimal. After the input voltage reaches
Figure 4.5: Effect of input voltage on the surface velocity of the oscillating diaphragm.

40 VAC, cavitation was observed on the surface of the diaphragm. For the frequency at which this experiment was conducted (6.99 kHz), the threshold surface velocity was reached at an input voltage of approximately 40 VAC. Above this level, increasing the voltage has minimal affect on the strength and sustainability of the jet.

Since the piezoelectric material used in this analysis was shown through experimentation to have an input voltage limit of approximately 120 VAC, the least amount of voltage possible should be used to form the jet. The more voltage fed to the driver, the shorter the duration that the driver will operate before cracking the piezoelectric material. Also, it should be noted that this threshold input voltage is only valid for the set-up used in this experimentation. By changing the inner diameter of the
tube onto which the driver is mounted, more input voltage may be required to reach the appropriate surface velocity. Since the surface velocity does not change very much once 48 VAC has been supplied to the driver, between 50 and 60 VAC should be a sufficient amount of input voltage for most applications.

Another area of interest in Figure 4.5 is the jump in surface velocity that occurs between 20 and 30 volts. Since this phenomenon did not occur when this experiment was conducted in air, it is conjectured that this jump is due to the phase of the fluid on the 'wet' side of the driver. For input voltages less than 30 VAC, a small bubble formed on the wet surface of the diaphragm. This bubble was forced off the surface of the diaphragm when the voltage increased to 30 VAC and the surface velocity increased by a factor of six to 1200 mm/s.

4.2 Pool Boiling Results

4.2.1 Optimum Height Determination

The effect of the distance between the driver and the thermal test die was investigated. As mentioned before, each of the shims had six equally spaced holes with the first one being at 7.6 mm and the last being at 26.7 mm. To set the separation distance between the diaphragm and the test die, the acrylic block was bolted to the desired holes in each of the shims and placed in the water tank, as shown in Figure 3.3. The jet and thermal test die were energized as described in Section 3.3.1. The steady-state heat flux dissipated by the die as a function of die temperature was recorded for each of the six height adjustments. In Figure 4.6, the effect of separation distance on the heat dissipated from the thermal test die is plotted. These results indicate that the best
Figure 4.6: Effect of separation distance between jet driver and thermal test vehicle. Separation distance = 1.9 mm (●), 5.7 mm (○), 9.5 mm (□), 13.3 mm (▲), 17.1 mm (△), and 20.9 (●).

The overall separation distance is 5.7 mm. Below a die temperature of 100°C, the 5.7 mm separation distance provided the best heat transfer results. At this distance the jet was close enough to increase mixing providing increased convection, but not too close to impede the jet performance. At the closest separation distance of 1.9 mm, the jet driver and its assembly interfered with the ability of the jet to diffuse the heat away from the die. Above a die temperature of 100°C, the three closest separation distances of 1.9, 5.7, and 9.5 mm provided similar heat flux results as a function of die temperature. At each of these distances, even though the level of mixing may have been different, the jet is capable of removing the vapor bubbles. The rewetting of the surface dominated the heat.
transfer process providing nearly identical heat flux results as a function of temperature. At separation distances larger than 10 mm, the jet momentum was not sufficient to remove all the vapor bubbles decreasing the heat transfer dissipation levels. The ability of the cavitation jet to dissipate over 110 W/cm² at a die temperature of 103°C with a driver to die separation distance of 1.9 mm indicates that a small-scale cavitation jet module could be produced for microelectronic thermal management.

4.2.2 Optimum Angle Determination

The purpose of this analysis was to quantify the effect that the impingement angle had on the heat dissipated from the heater’s surface. After determining the optimum separation distance between the diaphragm and the test die, the acrylic block was set to this height by bolting it to the appropriate holes on the shims. The angle of the diaphragm was established by rotating the acrylic block to the appropriate angle using the bolts as the point of rotation. For each angle, the placement of the center of the block was altered as to maintain a constant distance between the two surfaces. After placing the actuator assembly back into the water tank, the cavitation jet was tested as described in Section 3.3.2. Figure 4.7 shows the heat flux as a function of thermal test vehicle temperature for the four different impingement angles. The results plotted in Figure 4.7 indicate that jet angle of attack has little effect on the heat flux dissipated by the die, particularly above a die temperature of 100°C. In each of the angle configurations, the jet removed the vapor bubbles that formed on the die. Thus, the die experienced the same conditions for each angle resulting in nearly identical heat transfer results. The ability to vary the impingement angle of the jet provides further design flexibility in the
Figure 4.7: Effect of jet impingement angle. Jet impingement angle with respect to the horizontal = 0° (●), 15° (x), 25° (c), and 35° (▲).

implementation of a cavitation jet heat transfer module for microelectronic cooling applications.

4.2.3 Heat Removal Capability of Cavitation Jet

The purpose of this analysis was to measure the effect of the jet impinging directly on the thermal test die with the jet positioned at its optimum separation distance and impingement angle. In this analysis, temperature data was obtained from the thermal test die for two different scenarios: pool boiling without the jet actuated and pool boiling with direct impingement from the turbulent jet. The results in Figure 4.8 show the effect of the jet on the heat dissipated by the die. Below a die temperature of 100°C and the
onset of boiling, the presence of the jet increased the amount of heat dissipated for a given temperature. The turbulent jet provided forced convection heat transfer from the die to the pool of water. The added turbulent mixing and bulk motion present when the jet is operational compared to the free convection of heat when the jet is off resulted in higher heat fluxes. The results of the cavitation jet can be most noticed at surface temperatures above 100°C. Without the insulating vapor blanket present, the heat flux dissipated as a function of die temperature increased at a faster rate. At a die temperature of 115°C, the heat flux dissipated increased from 36 W/cm² with the jet off to 119 W/cm² when the jet was on. With the jet on, this yields a convective heat transfer coefficient with the jet actuated of 1.29 W/cm²K compared to 0.4 W/cm²K with the jet off. This
230% (with a conservative uncertainty of 1%) improvement was the result of the
cavitation jet increasing the overall heat transfer coefficient through increased convection
and removal of the insulating vapor blanket.

Photographs of an operational jet removing vapor bubble from the die set-up
described above are presented in Figure 4.9. The photographs are taken 0.02 sec apart
after the piezoelectric disk is energized from rest. These images indicate the ability of the
jet to rapidly force the vapor bubbles of various diameters from the die into the bulk
liquid where they condense. By operating the driver at one resonant frequency, the jet
forced the bubbles to coalesce and become unstable. The turbulent motion of the jet
further ejected the unstable bubbles from the thermal test die.

4.2.4 Reduced Pressure Analysis

The motivation for this research was to obtain a better understanding of the role
that ambient pressure plays in the formation and durability of the cavitation jet. As air
was removed from the test module, the cavitation jet underwent several different changes.
First of all, it was observed that the jet weakened as the pressure in the chamber was
reduced. The strength of the cavitation jet is directly linked to how tightly the cavitation
bubbles, from which it is formed, are clustered. As the pressure within the chamber
decreased, this cavitation bubble cluster became unstable, and as a result began spreading
across the surface of the diaphragm. As a result of this spreading, the cavitation jet began
changing directions and intensities. The noticeable effects of the cavitation jet ended
when the pressure in the chamber was reduced to approximately 0.34 bar (5 psia). Past
Figure 4.9: Photographs of cavitation jet induced bubble ejection.
this point, bulk fluid motion continued, but it was not concentrated, as was the cavitation jet. This bulk fluid motion appeared to be a result of a large low pressure region located directly below the vibrating diaphragm. This low pressure region formed a dome of cavitation bubbles beneath the diaphragm’s surface. This phenomenon can be seen in Figure 4.10a. At approximately 0.14 bar (2 psia), this dome region began growing rapidly as the entire test module filled with bubbles, as displayed in Figure 4.10b. All of the noticeable bulk fluid motion ended when the pressure in the chamber reached 0.05 bar (0.75 psia).

Another observed change in the cavitation jet at reduced pressures was the direction from which fluid was drawn to the cavitation bubble cluster. Before air was removed from the test module, the water ejected downward by the cavitation bubble cluster was drawn across the face of the driver in a direction that was perpendicular to the direction of the cavitation jet. Once air was removed from the test apparatus, water would flow to the cavitation bubble cluster at an angle between 0° and 90° with respect to

Figure 4.10: a) Visualization of low pressure dome at a pressure of 0.34 bar b) Visualization of ‘foaming’ phenomenon at a pressure of 0.14 bar.
the face of the driver. The exact angle was a function of the pressure within the chamber. Once the terminal pressure for the cavitation jet was reached, the flow to the cavitation bubble cluster was approximately perpendicular to the face of the driver.

4.3 Channel Boiling Results

4.3.1 Heat Removal Capability of Cavitation Jet Using Test Die as Heat Source

In the first experiment, the water bath was set such that the water it provided to the heat transfer cell remained at a constant 20°C, and the flowrate of the water bath was set such that the water flowed at a rate of 5 ml/s through the cell. Data was collected for two different scenarios: channel boiling with the jet on and channel boiling with the jet
off. The results of this experimentation can be seen in Figure 4.11. Without the jet actuated, the crossflow within the cell did little to remove the insulating vapor bubbles from the surface of the test die. Without the jet actuated, the heat flux from the die to the water was 36 W/cm² at a surface temperature of 111°C. With the jet actuated, the heat transfer was much more significant. Below a die temperature of 100°C (the onset of boiling) the presence of the jet increased the amount of heat dissipated for a given temperature. The turbulent jet provided additional forced convection heat transfer from the die to the flowing water within the cell. The added turbulent mixing and bulk motion present when the jet was on compared to the very mild amounts of forced convection heat transfer when the jet was off resulted in higher heat fluxes. Above 100°C, the cavitation jet removed the insulating vapor bubbles from the surface of the die. As a result of this, the heat flux dissipated as a function of die temperature increased at a faster rate. While the crossflow alone had little effect on the heat dissipated from the surface, the flow, coupled with the cavitation jet, worked well to sweep the newly discharged bubbles downstream where they were allowed to condense. In addition to this, the crossflow provided cooler water to the jet thus allowing for more heat transfer due to the forced convection by the jet on the heated surface. At a die temperature of 111°C, the direct impingement of the jet onto the heated surface increased the heat flux from the surface by 236% (with a conservative uncertainty of 1%) to a value of 121 W/cm². With the jet actuated, the convective heat transfer coefficient increased from 0.4 W/cm²K to 1.33 W/cm²K. This improvement was a result of the cavitation jet increasing the overall heat transfer coefficient through increased convection and the removal of the insulating vapor bubbles.

69
Figure 4.12: Effect of jet when inlet water temperature is 20°C (x) vs. effect of jet when inlet water temperature is 60°C (o).

4.3.1.1 Bath temperature influence on heat transfer performance

In the second experiment, the water bath was again set such that the velocity of the water traveling through the cell was approximately 5 m/s, but in this experiment, the temperature of the water provided by the water bath was varied from 20°C to 60°C. At both bath temperatures, data was collected for two different scenarios: channel boiling with the jet on and channel boiling with the jet off. The results of this experimentation can be seen in Figure 4.12. Increasing the water temperature decreases the heat transfer for both the convective range (T<100°C) and the boiling range (T>100°C). However, the improvement in heat transfer at the higher temperature is even greater compared to having the jet off. This is attributed to the fact that the strength of the jet remains
Figure 4.13: The effect of the separation distance on heat transfer enhancement. Separation distances: 2.5 mm (○), 5 mm (●), 7.5 mm (▲), 10 mm (×).

virtually constant at various bath temperatures, while the amount of heat dissipated during standard pool boiling decreases as the temperature of the water in the bath increases. Thus, the jet provided a greater overall improvement in heat transfer for the higher bath temperature. This is directly reflected in the convective heat transfer coefficient. At the higher bath temperature, the heat transfer coefficient is 1.87 W/cm²K compared to 1.45 W/cm² obtained at the lower temperature. This increase in heat transfer coefficient is a result of the increased conductivity of the water at the higher temperatures, which is exacerbated by the increased mixing provided by the jet.
4.3.2 Heat Removal Capability of Cavitation Jet Using Calibrated Heater

4.3.2.1 Optimum height determination

Figure 4.13 displays the influence that the separation distance between the surface of the calibrated heater and the surface of the driver has on the heat transfer capability of the cavitation jet. As was the case in the separation distance experimentation conducted in the pool boiling analysis, the optimum separation distance is approximately 5 mm. Above this separation distance, the efficacy of the cavitation jet for removing heat from the surface decreases with increasing distance. This result is similar to that observed in the pool boiling experimentation, but in the case of this analysis, the crossflow within the cell exacerbated the decline in performance because of its interference with the flow pattern of the jet.

One result that differed from that which was observed in the pool boiling experimentation is the performance of the cavitation jet in close proximity of the heated surface (less than 2.5 mm of separation distance). In the case of pool boiling, the performance of the cavitation jet declined in close proximity of the heated surface due to the heated water and vapor becoming trapped between the driver and the heated surface. By introducing a crossflow as was done in this set of experimentation, the heated water and vapor were swept downstream and away from the heated surface, even at minimal crossflow velocities. This led to a heat flux of approximately 260 W/cm² at a surface temperature of 150°C. This heat flux is slightly better than that which was recorded for the aforementioned 5 mm separation distance at the same surface temperature. The reason for selecting the 5mm separation distance as the optimal distance is due to the fact that both the 2.5 mm and the 5 mm cases yielded approximately the same results, but
Figure 4.14: The effect of inclining the cavitation jet heat transfer module on the point where the jet no longer affects the amount of heat dissipated from the calibrated copper heater. Baseline case with no cavitation jet (○), jet actuated with no incline (▲), jet actuated with cell inclined 45° (●).

with a slightly increased separation distance, the vapor leaving the heated surface would not impinge onto the surface of the driver. By avoiding this impingement, long-term surface damage to the driver could be minimized.

4.3.2.2 Effects of cell rotation on heat transfer results

For this experiment, the water bath was set such that the water it provided to the heat transfer cell remained at a constant 20°C, and the flowrate of the water bath was set such that the water flowed at a rate of 5 ml/s through the cell. The purpose of this test was to establish the cavitation jet’s efficacy for increasing the amount of heat dissipated from the surface of the calibrated heater at high surface temperatures. In addition to this,
experimentation was conducted in an attempt to lengthen the amount of time that the jet would influence the heat transfer from the surface. To accomplish this, the cavitation jet heat transfer module was rotated 45° such that the water inlet was located below the water outlet. For both cases, data was collected for two different scenarios: channel boiling with the jet on and channel boiling with the jet off. The results of this experimentation can be seen in Figure 4.14. The first case examined was the uninclined case. When the surface temperature of the calibrated heater was approximately 120°C, the influence of the cavitation jet on the heat dissipated from the heater’s surface was at its maximum. As the surface temperature of the calibrated heater approached 150°C, the influence of the cavitation jet decreased to a point where the cavitation jet had no influence on the heat dissipated from the surface of the heater. By inclining the cavitation jet heat transfer module and the calibrated heater 45° such that the water inlet was located below the water outlet, the influence of the cavitation jet did not become minimal until a much higher surface temperature was achieved. Because of this, a heat flux of over 350 W/cm² was attainable before the jet no longer influenced the heat dissipation from the surface of the heater. By inclining the heat transfer module, the bubbles were naturally more inclined to move upward toward the outlet of the cell, particularly at the higher surface temperatures. Because of this added assistance, the jet was able to continue removing bubbles from the heated surface at temperatures higher than what was attainable without inclining the module. With the module uninclined, the jet was strong enough to expel the heated water from the heated surface in all directions, including upstream of the heater. Because of this, the jet was provided with increasingly warmer water to eject toward the heated surface. By inclining the cell, the heated water being
Figure 4.15: Heat dissipation capabilities of channel boiling heat transfer cell with the cavitation jet activated and a crossflow rate of: 0 ml/s (x), 1.5 ml/s (○), 3.5 ml/s (△), 5.5 ml/s (□), and 7.5 ml/s (★).

 expelled from the heater remained almost entirely downstream of the heater thus cooler water was supplied to the jet. Because of this, the overall heat transfer enhancement potential of the cavitation jet increased as the angle of the cell increased.

4.3.2.2 Effect of flowrate on heat transfer

To optimize the performance of a heat transfer cell with an induced crossflow, it is important to characterize the optimal crossflow rate such that the maximum heat transfer performance is obtained. One would assume that the heat transfer from the surface would continue to increase with increasing crossflow velocity, but since this experimentation is utilizing the cavitation jet in addition to the crossflow, the interference
with the jet must be considered. In each of these experiments, the cavitation jet is activated, and the crossflow rate is varied. The results of this experimentation can be seen in Figure 4.15. With no flow, the confined geometry of the cell caused the water within the cell to heat to approximately 70°C when approximately 100 V was supplied to the heater. This led to critical heat flux, 240 W/cm², being reached at a surface temperature of 140°C. By supplying even a minimal amount of flow through the cell, critical heat flux was not reached before the capacity of the experimental apparatus was reached. In this experimentation, the lowest flowrate tested was approximately 1.5 ml/s, which caused the average water temperature within the cell to be approximately 40°C at 100 V of input power. At this flowrate, the boiling curve was very similar to that which occurs with no flow with the major difference being that critical heat flux was not reached. By increasing the flowrate, the heat transfer improvement capabilities of the jet declined, because the flow interfered with the formation and stability of the cavitation jet.

4.3.3 Comparison of Cavitation Jet to Single Round Gas Jet

Self-similarity of a conventional round jet implies that its characteristic radius and the inverse of its centerline velocity increase linearly with distance from the jet orifice [49]. Figure 4.16 shows that the streamwise variation of jet radius and inverse velocity vary linearly with increasing distance from origin indicating that the mean flow of the cavitation jet is indeed self-similar. These values were measured by examining high-speed video of the cavitation jet for a known frame rate. Extrapolation of the jet radius from Figure 4.16 yields a characteristic jet diameter of 2.2 mm which is commensurate with the diameter of the cavitation bubble cluster as reported by James [6].
Figure 4.16: Variation of the inverse of centerline velocity (\(x\)) and jet half-width radius (\(\cdot\)) with distance from diaphragm.

In order to calculate the Reynolds number of the cavitation jet, the virtual origin was first established. From Figure 4.16, the virtual origin is the point where the inverse velocity curve intersects the \(x\)-axis, which occurs at approximately \(-1.14\) cm. It is from this point that the Reynolds number was calculated using the similarity function for a round turbulent jet, see White [49].

\[
U_{\text{cr}} = 7.4 \frac{(J/p)^{y^2}}{y} \tag{4.1}
\]

where \(y\) is the distance between a point within the streamwise flow of the jet and the virtual origin. The momentum flux, \(J\), for a uniform flow out of an equivalent round nozzle is
Figure 4.17: $yU_{cl}$ vs. $y$

\[ J = \frac{\pi}{4} d^2 \rho V_i^2 \]  \hspace{1cm} (4.2)

and the Reynolds number is

\[ Re = \frac{dV}{\nu} = \frac{2}{\sqrt{\pi}} \frac{\sqrt{J/\rho}}{\nu} \]  \hspace{1cm} (4.3)
where \( V_e \) is the exit velocity of the jet and \( d \) is the characteristic diameter of the jet nozzle. By combining Equations 4.1 and 4.2, the Reynolds number for a cavitation jet can be expressed as

\[
Re = \frac{2 (\nu U_{cl})}{7.4\sqrt{\pi} \nu} \quad .
\]  

From Figure 4.17, \( x_0 U_{cl} \) is approximately constant and its value is 95 cm/s. From Equation 4.4, the Reynolds number of the cavitation jet is approximately 1450.

Using the parameters yielded from Figure 4.16, Equation 4.5 was used to calculate the Nusslet number based on the heat transfer obtained while using the cavitation jet.

\[
Nu = \frac{hd}{k} \quad .
\]

From the pool boiling experimentation, the heat transfer coefficient, \( h \), is approximately 12900 W/m²K and the conductivity of water at 20°C is 0.68 W/mK. From Equation 4.5, the experimental Nusselt number is 42.

As proposed by Martin [50], the following correlation can be used to estimate the Nusselt number of a single round gas jet.

79
\[
\frac{Nu}{P_e^{1/2}} = G \left( \frac{R}{d} \right) \left( \frac{Z}{d} \right) F_i(Re)
\]  
(4.6)

where

\[
F_i = 2 Re^{1/2} \left( 1 + 0.005 Re^{0.5} \right)^{1/2}
\]  
(4.7)

and

\[
G = \frac{d}{R} \frac{1 - 1.1d/R}{1 + 0.1(Z/d - 6)d/R}
\]  
(4.8)

For the cavitation jet, the diameter, \(d\), of the jet at its origin is approximately 2.2 mm, the distance, \(R\), represents the half distance of the heated surface onto which the jet is impinging which is approximately 5 mm, the separation distance, \(Z\), between the heated surface and the surface of the diaphragm is 5 mm, the approximate Reynolds number of the jet is 1450, and the approximate Prandtl number of the water at 293 K is approximately 7. From Equation 4.6, the estimate of the Nusselt number of the jet is 53.

By comparing the Nusselt numbers of the cavitation jet with the one obtained from the correlation for the traditional round jet, it was determined that the cavitation jet provides less heat transfer improvement than a traditional round turbulent jet with the same parameters.
CHAPTER 5

REPRODUCIBILITY AND ERROR

5.1 Error Analysis

This section provides the sources of various measurement errors that occurred in the experimentation conducted for this analysis. The measurement uncertainties presented in this section are based on the assumption that all measured quantities follow a symmetric distribution and that error limits are twice the standard deviation. In addition, an instrument with an error of ±α will have a standard uncertainty \( u = \alpha / 2 \).

5.1.1 Standard Lab Equipment Error

Inherent within all laboratory equipment is a resolution that defines to what accuracy measurements with that equipment can be obtained. Table 5.1 displays the uncertainties for some of the equipment used when conducting this analysis.

<table>
<thead>
<tr>
<th>Equipment</th>
<th>Standard Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>Caliper</td>
<td>0.013 mm</td>
</tr>
<tr>
<td>Protractor</td>
<td>0.25°</td>
</tr>
<tr>
<td>Function Generator</td>
<td>0.005 kHz</td>
</tr>
<tr>
<td>Multimeter</td>
<td>0.0005 V</td>
</tr>
<tr>
<td>Flow Meter</td>
<td>0.025 gpm</td>
</tr>
</tbody>
</table>
5.1.1.1 Error in pressure transducer measurements

The error associated with the pressure transducer is associated not only with the uncertainty of the measurements taken by the transducer but also the uncertainty associated with the multimeter used to display the output. With the uncertainty of the pressure transducer being 0.14 mV, as described by the manufacturer and the uncertainty of the multimeter being 0.0005 V, Equation 5.1 displays the uncertainty of the reduced pressure measurements.

\[ \sigma_{\text{press}}^2 = \sigma_{\text{trans}}^2 + \sigma_{\text{mult}}^2 \]  

(5.1)

Therefore, the uncertainty of the pressure measurements is 0.52 mV or 9.3\times10^{-5} \text{ psi}.

5.1.1.2 Error in surface displacement and acceleration calculations

The error associated with the surface displacement and acceleration calculations is directly related to the error associated with the vibrometer. Within the vibrometer, there are many sources of error, but all are negligible, including the manufacturer’s error estimate, when compared to the error associated with the signal-to-noise ratio. Through trial and error, it was determined that an uncertainty of 6 mV is conservative enough to envelope all of the minute errors as well as the error associated with the signal-to-noise ratio. Even though the data collected from the vibrometer was recorded from a multimeter, the uncertainty of the multimeter is negligible when compared to that of the
vibrometer itself. From Equation 3.1, the uncertainty associated with the peak-to-peak velocity is given by Equation 5.2.

\[ u_{v_{max}} = \left( \frac{\partial (2x\sqrt{2v_{max}})}{\partial v_{max}} \cdot u_{v_{max}} \right)^2 \]  

(5.2)

Given that the correlation coefficient, \( x \), is 125 mm/s/\( \sqrt{\text{m/s}} \), the standard uncertainty of the velocity is 2.12 mm/s. From Equation 3.2, the uncertainty of the displacement can be obtained by evaluating Equation 5.3 at a nominal frequency, \( f \), of 9 kHz and a conservative velocity, \( \text{Vel}_{\text{cons}} \), of 3900 mm/s.

\[ u_d^2 = \left( \frac{\partial (\text{Vel}_{\text{cons}}/2\pi f)}{\partial \text{Vel}_{\text{cons}}} \cdot u_{\text{Vel}_{\text{cons}}} \right)^2 + \left( \frac{\partial (\text{Vel}_{\text{cons}}/2\pi f)}{\partial f} \cdot u_f \right)^2 \]  

(5.3)

The standard uncertainty of the surface displacement is 40 nm, or approximately 0.24 percent. From Equation 3.3, the uncertainty of the acceleration can be obtained by evaluating Equation 5.4 at a nominal frequency, \( f \), of 9 kHz and a conservative displacement, \( \delta \), of 0.017 mm.

\[ u_a^2 = \left( \frac{\partial (2\pi f^2 \delta)}{\partial \delta} \cdot u_{\delta} \right)^2 + \left( \frac{\partial (2\pi f^2 \delta)}{\partial f} \cdot u_f \right)^2 \]  

(5.4)

83
The standard uncertainty of the surface acceleration is 60,000 mm/s², or approximately 0.26 percent.

5.1.2 Thermal Test Die

The surface temperature of the die was assumed to be equal to the internal temperature readings from the diodes buried within the chip. Therefore, the uncertainty of the temperature data collected from the die is a function of the uncertainties associated with measurements taken by the diodes as well as the uncertainties associated with the Fluke NetDAQ data acquisition system. Given that the uncertainty of a four-wire diode is 0.5°C, and the uncertainty of the data acquisition system is also 0.5°C, the uncertainty of the temperature measurements is as shown in Equation 5.5.

\[ u_T^2 = u_{\text{diode}}^2 + u_{\text{DAQ}}^2 \]  \hspace{1cm} (5.5)

Therefore, the uncertainty of the temperature measurements is 0.71°C. In order to calculate the uncertainties inherent within the calculation of the power provided to the die, the following equation was used.

\[ u_P^2 = \left( \frac{\partial(VI)}{\partial V} u_I \right)^2 + \left( \frac{\partial(VI)}{\partial I} u_V \right)^2 \]  \hspace{1cm} (5.6)

84
where the uncertainty of the voltage measurements are 0.0005 V and the uncertainty of the current measurements, \( u_i \), are 0.0005 A. The uncertainty of the heat transfer measurements was found for a voltage of 70 V and a current of 2.5 A as these values would yield a conservative value for error. From these values, \( u_q \) is 0.035 W.

5.1.3 High Heat Flux Surface

Since the calibrated heater used in this analysis was the exact one used by Sellers [1] in performing her research, the following text was taken directly from her thesis with the appropriate changes made to reflect modifications made to suit the needs of this analysis. Uncertainties in the calculation of the surface temperature and the heat flux for the high heat flux surface are generated from errors associated with the measurements of the thermocouples as well as their location within the neck region of the copper block.

5.1.3.1 Surface temperature

The surface temperature, \( T_s \), is calculated by extrapolating the temperature gradient between thermocouple levels 1 and 2 as shown in Figure 5.17. This expression can be written as

\[
T_s = T_1 - \left( \frac{\Delta T}{\frac{A_{c1} - A_{c2}}{A_1} + \frac{A_{c2}}{A_2}} \right)
\]

(5.7)

where the average temperature at location 1 is \( T_1 \) and the difference between the average temperatures at locations 1 and 2, \( \Delta T \), can be written as

85
\[ \Delta T = T_i - T_1 \]  

(5.8)

Geometrical dimensions are shown in Figure 5.17 and include: \( \Delta z \), the axial distance between the first two thermocouple levels; \( \Delta z_{c-s} \), the distance between the upper level of thermocouples and the surface; \( \Delta z_s \), the thickness of the indented surface region; \( A_{TC} \), the cross-sectional area of the neck region having a width of \( w \); and \( A_s \), the surface area having a width of \( w_s \).

From Equation 5.7, it can be seen that the uncertainty in the extrapolated surface temperature results from two sources: inaccuracies in geometrical dimensions and errors associated with the thermocouple measurements. The uncertainty for the surface temperature can be determined by Equation 5.9.

\[
\begin{align*}
\sigma^2 &= \left( \frac{\partial T_1}{\partial T_i} \sigma_{T_i} \right)^2 + \left( \frac{\partial T_1}{\partial \Delta z} \sigma_{\Delta z} \right)^2 + \left( \frac{\partial T_1}{\partial \Delta z_c} \sigma_{\Delta z_c} \right)^2 + \left( \frac{\partial T_1}{\partial A_{TC}} \sigma_{A_{TC}} \right)^2 \ldots \\
&\ldots + \left( \frac{\partial T_1}{\partial w_s} \sigma_{w_s} \right)^2 + \left( \frac{\partial T_1}{\partial A_s} \sigma_{A_s} \right)^2
\end{align*}
\]  

(5.9)

The average temperature at level 1, \( T_i \), is determined from the three thermocouples at this location. The uncertainty of this average temperature can be determined from the following

\[ \sigma^2 = \frac{1}{3} \sigma_{T_{TC}}^2 \]  

(5.10)
where the uncertainty of an individual thermocouple measurement, $u_{TC}$, is a function of the accuracy of both the thermocouple wire and the data acquisition system.

\[ u_{TC}^2 = u_{w}^2 + u_{DA}^2 \] (5.11)

According to the distributor of the thermocouple wire, the limits of error for the T-type wire are ±0.5°C. The limits of error associated with the data acquisition system are ±0.5°C for T-type thermocouples. Therefore, the uncertainty of an individual thermocouple measurement is 0.707°C. From equation 5.10, the uncertainty in the average temperature at one axial location is 0.408°C.

Similarly, the uncertainty in the difference between the average temperature at two locations can be determined by the following expression.

\[ u_{AT}^2 = u_{t1}^2 + u_{t2}^2 + 2u_{r}^2 \] (5.12)

Substituting Equation 5.10 into this expression results in

\[ u_{AT} = u_{TC} \sqrt{\frac{2}{3}} \] (5.13)

Therefore, the uncertainty of the calculation of the axial temperature difference is 0.58°C.

The thermocouples are assumed to be centered within a hole of diameter 1.32 mm to an accuracy of 5 percent of the hole diameter. This is a valid assumption since the
hole was only slightly larger than the ceramic insert and the thermocouple junction was centered in the ceramic insert. Therefore, the uncertainty between the top thermocouple location, level 1, and the surface represented by \( u_{a_{1}-s} \) is simply the error associated with centering the thermocouple junction within the hole.

\[
u_{a_{1}-s} = u_s = (1.005)(1.32 \text{ mm}) = 0.066 \text{ mm} \tag{5.14}
\]

The uncertainty of the spacing between the two levels of thermocouples, \( u_{a_{1}} \), can be expressed by the following

\[
u_{a}^{2} = u_{i}^{2} + u_{s}^{2} = 2u_{s}^{2} \tag{5.15}
\]

or

\[
u_{a_{1}} = \sqrt{2} u_s \tag{5.16}
\]

Again assuming an error of 5 percent in locating the thermocouple junction in the center of the hole, the uncertainty in the axial distance between thermocouple levels, \( u_{a_{1}} \), is 0.0934 mm.

The remaining three uncertainties involve the accuracy in measurement of the width of the surface, \( w_s \), the width of the neck region, \( w \), and the thickness of the surface region, \( \Delta z \), as shown in Figure 5.17. The limits of error associated with the
measurement of widths $w$ and $w_i$ are assumed to be $\pm 0.2$ mm. Calculation of the uncertainties in the surface area can be determined from the following equations, where the width of the surface was 8.0 mm and the width of the neck region was 10.0 mm.

$$a_{u_w}^2 = \left( \frac{\partial A}{\partial w} u_w \right)^2 = (2w u_w)^2$$  \hspace{1cm} (5.17)

$$a_{u_i}^2 = \left( \frac{\partial A}{\partial w_i} u_{w_i} \right)^2 = (2w_i u_{w_i})^2$$  \hspace{1cm} (5.18)

The resulting uncertainty of the surface area, $A$, is 3.2 mm$^2$ and 4.0 mm$^2$ for the cross-sectional area for the neck region, $A_{TC}$. A measurement of 1.5875 $\pm$ 0.1 mm is assumed for the thickness of the surface region, $\Delta z$. Assuming a temperature difference of 10°C between the thermocouple levels, the surface temperature can be determined to an uncertainty within 1.00°C.

5.1.3.2 Heat flux

A similar analysis can be performed on the calculation of the heat flux. The heat flux can be calculated by assuming one-dimensional heat flux in the neck region of the heater. Using Fourier’s Law

$$q = -\frac{A_{TC} \Delta T}{\Delta z}$$  \hspace{1cm} (5.19)
where the temperature difference can be defined as

\[ \Delta T = T_2 - T_1 \] (5.20)

and the axial distance between thermocouple levels is \( \Delta z \). The surface area of the heater, \( A_s \), and the cross-sectional area of the neck region, \( A_{NC} \), are known, as well as the thermal conductivity, \( k \), of the heater block. Therefore, the uncertainty of the heat flux calculation can be evaluated

\[ u_q^2 = \left( \frac{\partial q}{\partial k} u_k \right)^2 + \left( \frac{\partial q}{\partial \Delta T} u_{\Delta T} \right)^2 + \left( \frac{\partial q}{\partial \Delta z} u_{\Delta z} \right)^2 + \left( \frac{\partial q}{\partial A_s} u_{A_s} \right)^2 + \left( \frac{\partial q}{\partial A_{NC}} u_{A_{NC}} \right)^2 \] (5.21)

The conductivity of the copper heater block was assumed to be a constant value of 3.922 W/cm°C. Over the temperature range investigated, 110°C to 130°C, the thermal conductivity varied by less than 0.009 W/cm°C, or 0.22 percent [48]. Uncertainties for the remaining variables in Equation 5.21 have been previously discussed in Section 5.1.3.1. For a typical temperature difference, \( \Delta T \), of 10°C, the calculated heat flux from Equation 5.19 is 193 W/cm² with an uncertainty of 16.7 W/cm² or 8.6 percent.

5.1.3.3 Error associated with thermocouple holes

The previous sections discussed the uncertainties due to the inaccuracies in the measuring equipment. However, a second source of error could be introduced by the presence of the actual measuring device. For this investigation, the insertion of the
thermocouples into holes in the neck of the heater altered the heat flux in the measurement area. As discussed in Section 3.5, three thermocouples were inserted into 1.32 mm diameter holes at each axial location, all at a depth of 2.54 mm. These holes caused a decrease in the cross-sectional area of the metal at the measurement location. Since the heat transfer rate through the neck was assumed to remain constant, the local temperature measured by a thermocouple at the reduced cross-sectional area must increase. Thus, the thermocouples measured an inflated temperature difference leading to errors in the determination of heat flux and surface temperature. A conservative estimate of this error can be made by assuming that no heat is conducted through the thermocouple holes. The measured heat flux between two axial locations can be determined by the following equation

\[
q_{\text{meas}} = -k \frac{\Delta T_{\text{meas}}}{\Delta z} \tag{5.22}
\]

The measured temperature difference, \(\Delta T_{\text{meas}}\), is the sum of the actual temperature difference, \(\Delta T_s\), and the temperature difference across the neck region containing the holes, \(\Delta T_{\text{holes}}\), minus the temperature difference across that same region if the holes did not exist, \(\Delta T_{\text{solid}}\).

\[
\Delta T_{\text{meas}} = \Delta T_s + 2 \left( \frac{\Delta T_{\text{holes}}}{2} - \frac{\Delta T_{\text{solid}}}{2} \right) = \Delta T_s + \Delta T_{\text{holes}} - \Delta T_{\text{solid}} \tag{5.23}
\]

Substituting into Equation 5.22

91
\[ q_{\text{mean}} = -k \frac{\Delta T_r + \Delta T_{\text{hole}} - \Delta T_{\text{wall}}}{\Delta z} \]  

(5.24)

The actual heat flux can be expressed as the following

\[ \Delta T_r = \frac{-q_{\text{mean}} \Delta z}{k} \]  

(5.25)

If the holes did not exist, the temperature difference across this axial distance of \(2r\) would simply be

\[ \Delta T_{\text{wall}} = -\frac{q_{\text{mean}} 2r}{k} \]  

(5.26)

The heat transfer rate through the region containing the thermocouple holes must be equal to the rate through the region without holes, since one-dimensional conduction is assumed.

\[ q_s = \left( \frac{A(z)}{A_{\text{wall}}} \right) \left( -k \frac{dT}{dz} \right) \]  

(5.27)

The cross-sectional area at a given axial location, \(A(z)\), can be expressed as

92
\[ A(z) = w^2 - \left[ 2 \sqrt{r^2 - z^2} \right] \]

where the width of the square neck region, \( w \), the hole radius, \( r \), and the depth of the three holes, \( t \), are known. Substituting Equation 5.28 into Equation 5.27 and integrating across the diameter of the hole gives:

\[
\frac{7}{8} \int_{c}^{r} \frac{dT}{\tau} = - \frac{q_{c} A_{tot}}{k} \int_{c}^{r} \frac{1}{w^2 - \left[ 2 \sqrt{r^2 - z^2} \right]} \, dz
\]

where \( r \) and \( z \) are both in [cm], \( A_{tot} \) is in [cm\(^2\)], \( k \) is in [W/cm°C], and \( q_{c} \) is in [W/cm\(^2\)].

This expression can be analytically solved by integrating with the appropriate dimensions: \( w = 1 \) cm; \( r = 0.066 \) cm for the radius of the thermocouple holes; and \( t = 0.254 \) cm for the thermocouple hole depth. The result is:

\[
\Delta T_{tot} = -0.143 \frac{q_{c} A_{tot}}{k}
\]

Substituting Equations 5.25, 5.26, and 5.30 into 5.24 results in the following expression.

\[
\dot{q}_{\text{tot}} = - \frac{k}{\Delta z} \left[ - \frac{q_{c} \Delta z}{k} + \frac{(-0.143 q_{c} A_{tot})}{k} - \left( - \frac{q_{c} (2x)}{k} \right) \right]
\]

or

93
\[ q'_{\text{max}} = \frac{q_a}{\Delta z} (\Delta z + 0.143A_{\text{solid}} - 2r) \]  

(5.32)

For the heater in Figure 5.17, the axial distance between thermocouple levels, \( \Delta z \), is 3.175 mm, the area of the solid, \( A_{\text{solid}} \), is 100 mm\(^2\), and the radius of the thermocouple holes, \( r \), is 0.66 mm. Therefore, the measured heat flux can be expressed in terms of the actual heat flux.

\[ q'_{\text{max}} = 1.035q_a \]  

(5.33)

Therefore, the measured heat flux is 3.5 percent greater than the actual surface heat flux as a result of thermocouple measurement errors.

### 5.2 Reproducibility

Reproducibility is one of the major downsides associated with the cavitation jet technology. When producing the cavitation jet, all one can do is provide a signal and a current to the driver such that the driver oscillates above a given threshold. Outside of this, the formation of the jet is independent of human intervention. Because of this, producing a statistically identical jet repeatedly is essentially impossible.

As previously discussed, there are many factors that determine the strength and stability of the jet. Even if several jet producing apparatuses are built identically, each one will produce a jet with slightly different characteristics. There are many causes for this variance. First of all, small discrepancies in the piezoelectric driver can greatly
impact the performance of the jet. Since the drivers are each produced as part of a bulk, it is quite common to find disks where the ceramic material was not centered upon being glued to its surface. Secondly, repeated use of the disks causes pitting to occur on their surfaces. Once pitting begins, the useful life of the disk is over. Also, by leaving the disks submerged, its brass surface would tarnish, rendering the disk useless. These factors are only a few of many possible ones that can affect how the piezoelectric disk generates the cavitation jet.

In the short term, the jet is capable of producing fairly consistent results. Figure 5.1 shows an overlap of two different tests run using the cavitation jet. Both tests were conducted independently in the channel boiling cell using the same driver, and the difference between most of the pairs shown fall within the error of the analysis. These similar results do not occur after the driver has been extensively used.

![Figure 5.1: Reproducibility of cavitation jet for heat removal in channel boiling heat transfer cell.](image)
CHAPTER 6

CONCLUSIONS AND RECOMMENDATIONS

In this analysis, the efficacy of the cavitation jet for heat transfer enhancement was demonstrated. The cavitation jet was formed from a cluster of cavitation bubbles that are the result of a submerged piezoelectric diaphragm oscillating above a given surface velocity threshold. The cavitation jet was positioned in such a way as to impinge onto a submerged heated surface. By impinging the heated surface, the vapor bubbles that would normally cling to the surface during nucleate boiling were removed from the surface and were replaced with cool water from the bulk within the test module. In this analysis, two different heaters operating in two different flow environments were examined. For each heater in each environment, the cavitation jet significantly increased the heat transfer from the heated surface.

6.1 Variables Determining Jet Formation

There are many factors that dictate how and when the cavitation jet is formed. First and foremost, to generate a jet from the cavitation bubbles formed on the surface of a submerged piezoelectric diaphragm, the vibrating surface must exceed a threshold surface velocity. This threshold surface velocity was found to be 1100 mm/s. To achieve this threshold, there are many factors that must be considered.
Through this analysis, it was demonstrated how the resonant frequency of the diaphragm increases when the inner diameter of the holder onto which it is mounted decreases. It was determined that below an inner diameter of approximately 14 mm, the diaphragm can no longer achieve the surface velocity necessary for jet formation at its respective resonant frequency. In addition to the holder's inner diameter, another key variable is the temperature of the water bath into which the jet is submerged. As the temperature of the bath was increased, the surface velocity, for a given frequency and input voltage, decreased. It was shown that when the temperature of the driver is above 50°C, the surface could no longer achieve the surface velocity necessary for jet formation at said frequency and input voltage. To remedy this, the frequency of the diaphragm can be lowered or the input voltage supplied to the driver could be increased.

6.2 Jet Impingement in a Quiescent Pool

By impinging a submerged heated surface with the cavitation jet, heat transfer fluxes of up to 135 W/cm² were achieved in a compact geometry requiring approximately 1 W of input power. The tests reported here quantify the effectiveness of the cavitation jet in both atmospheric as well as low pressure environments. As shown, it is possible to generate a water jet from a cluster of cavitation bubbles with enough momentum to remove vapor bubbles from the surface of a submerged heated surface.

The present work has shown that the performance of the cavitation jet in terms of the heat flux for a given surface temperature increases as the separation distance between the heated surface and the diaphragm decreases. It was found that the optimal separation
distance between the diaphragm and the heated surface is approximately 5.7 mm. Comparative tests using different impingement angles indicate relatively no difference in the heat flux dissipation as a function of angle. For a surface temperature of 120°C, the heat flux increases 230% when the jet is energized when compared to traditional pool boiling. With the jet activated, the convective heat transfer coefficient is approximately 1.29 W/cm²K compared to 0.4 W/cm²K with the jet off. These heat transfer results indicate the potential of using the cavitation jet to effectively manage heat removal from high power microelectronics contained within confined geometries.

Reduced pressure analyses indicate that creating a cavitation jet is possible at pressures as low as 0.34 bar. Below this pressure, bulk fluid motion continues but not in a concentrated form. This bulk fluid motion appeared to be a result of a large low pressure region located directly below the vibrating diaphragm. This low pressure region formed a dome of cavitation bubbles beneath the diaphragm's surface. At approximately 0.14 bar, this dome region began growing rapidly as the entire test module filled with bubbles. All of the noticeable bulk fluid motion ended when the pressure in the chamber reached 0.05 bar. The cavitation jet is most effective once the temperature of the heated surface has surpassed the saturation temperature at the given pressure. Since the cavitation jet is capable of operating below atmospheric pressure, the jet can enhance the heat transfer from surfaces at reduced pressures where boiling begins below the nominal boiling temperature of 100°C. The ability to operate at low pressures is critical when utilizing computer chips as the heated surface as they cannot reach the standard boiling temperature without significant damage.
6.3 Jet Impingement in a Channel Flow Environment

In this experimentation, the cavitation jet was used to remove the vapor bubbles that formed on two different heaters during nucleate boiling and force them back into the cooler bulk liquid where the crossflow within the cell swept them downstream. For a surface temperature of 120°C, heat fluxes up to 165 W/cm² were achieved in a compact geometry requiring approximately 1 W of input power. This is a heat flux increase of 236% compared to traditional channel boiling with no jet, for the same surface temperature. At a bath temperature of 20°C with the jet actuated, the convective heat transfer coefficient increases from 0.4 W/cm²K to 1.33 W/cm²K. In the case of pool boiling, the heat transfer coefficient was 1.29 W/cm²K when the jet was actuated, which is 3% less than the same case in the channel flow cell. It was also determined that by increasing the temperature of the water flowing through the cell from 20°C to 60°C, the percent improvement in heat dissipation resulting from impingement by the cavitation jet increased to 300%. When the bath temperature is 60°C and the jet is actuated, the heat transfer coefficient is 1.87 W/cm² at a surface temperature of 115°C. This is a 29% improvement in heat transfer coefficient over the case where the water was 20°C. This increase in heat transfer coefficient is a direct result of the increased conductivity of the water at the higher temperature.

Similar to the pool boiling experimentation, the optimal separation distance between the surface of the driver and the heated surface is approximately 5 mm. The major difference that was noticed with the induced crossflow was the fact that the jet's
performance did not degrade at separation distances on the order of 1 to 2 mm as it did in the pool boiling experimentation. In the case of pool boiling, the performance of the cavitation jet declined in close proximity of the heated surface due to the heated water and vapor becoming trapped between the driver and the heated surface. By introducing a crossflow as was done in this set of experimentation, the heated water and vapor were swept downstream and away from the heated surface, even at minimal crossflow velocities.

In addition to the separation distance, another important parameter that was analyzed was the flowrate of water through the cell. With no flow, the confined geometry of the cell caused the water within the cell to heat to approximately 70°C when approximately 100 V was supplied to the heater. This led to critical heat flux, 240 W/cm², being reached at a surface temperature of 140°C. By supplying even a minimal amount of flow through the cell, critical heat flux was not achieved before the capacity of the experimental apparatus was reached. In this experimentation, the lowest flowrate tested was approximately 1.5 mL/s, which caused the average water temperature within the cell to be approximately 40°C at 100 V of input power. At this flowrate, the boiling curve was very similar to that which occurs with no flow with the major difference being that critical heat flux was not reached. By increasing the flowrate, the heat transfer improvement capabilities of the jet declined, because the flow interfered with the formation and stability of the cavitation jet. Another reason for this decline could be that the crossflow might have pushed the point of maximum heat transfer downstream of the heated surface. By utilizing a PIV system, future analyses could be conducted to confirm
or refute this claim. Based on the results of this analysis, the best possible flowrate is the smallest one tested or 1.5 m/s.

When using the calibrated heater as the heat source, it was shown that the greatest amount of improvement in heat flux for a given surface temperature occurred at 120°C. Beyond this point, the effectiveness of the jet declined until the surface temperature of the heater reached 150°C at which the amount of heat dissipated from the surface was approximately 300 W/cm². At this point, the jet no longer had any effect on the heat transfer from the calibrated heater. By inclining the heat transfer module and calibrated heater 45° such that the water inlet was positioned below the water outlet, the influence of the cavitation jet did not become minimal until a much higher surface temperature was achieved. Because of this, a heat flux of over 350 W/cm² was attainable before the jet no longer influenced the heat dissipation from the surface of the heater.

6.4 Jet Performance Compared to that of Traditional Round Jet

To obtain the Reynolds number of the cavitation jet used in this analysis, a similarity solution was utilized. After calculating the virtual origin of the cavitation jet, the Reynolds number was found to be 1450. In order to quantify the effectiveness of the cavitation jet, its Nusselt number was compared to that of a traditional turbulent round jet. Using the parameters established from the similarity solution coupled with the heat transfer coefficient obtained for pool boiling, the experimental Nusselt number of the cavitation jet was found to be 42. Using a correlation established by Martin [50], the Nusselt number of a traditional turbulent round jet was found to be 53. By comparing the

101
Nusselt numbers of the cavitation jet with the one obtained from the correlation for the traditional round jet, it was determined that the cavitation jet provides less heat transfer improvement than a traditional round turbulent jet operating under the same parameters.

6.5 Recommendations for Future Research

The results obtained from the research in this analysis adequately demonstrate the possibilities inherent within the cavitation jet technology. The research conducted in preparation for this thesis only scratches the surface of the research that must be conducted in order to fully understand why the cavitation jet is formed and how it can best be used. Future research should be conducted in order to better understand how the cavitation jet forms from the cavitation bubbles on the vibrating diaphragm. Once the formation of the jet is fully characterized, efforts should be focused on modeling the jet so that accurate predictions can be made on the ability of the cavitation jet to increase heat transfer from surfaces of various geometries submerged in containers of various sizes with varying flow characteristics. Also, future research should explore the possibility of using working fluids other than water to produce the jet. If the jet can indeed be created and maintained in a fluid with a lower vapor pressure, cooling of heat sensitive components could be possible without forcibly lowering the pressure within the heat transfer cell.

Significant efforts should focus on dampening the sound emitted by the vibrating diaphragm when creating the cavitation jet. Because the diaphragms used in this experimentation were forced to oscillate at high frequencies, the sound emitted by a
diaphragm was deafening. In order for the cavitation jet to be utilized in any facet outside of the lab, this sound needs to be substantially dampened. Also, during testing, the surface of the diaphragm experienced considerable pitting due to the cavitation bubbles on its surface. Future efforts should focus on either finding a harder diaphragm that has the same capacity for oscillation as the brass ones used in this experimentation, or efforts should focus on retrofitting the surface of the brass diaphragms with a small granularity of a denser metal such that the cavitation bubbles form on this material instead of on the brass itself.

Last, but certainly not least, future efforts should focus on fully identifying the capacity of the cavitation jet for heat removal. To do this, many modifications to the experimental apparatuses used in this research are necessary including modification of the heated surface. Literature research showed that by creating additional nucleation sites on the heaters surface, more bubbles form at the onset of nucleate boiling thus allowing the jet more of an opportunity to remove heated vapor from the heater’s surface. Also, it could be worthwhile to explore what happens when the working fluid itself is seeded as to, again, provide more nucleation sites within the heat transfer module. Most importantly, critical heat flux values need to be characterized in each set of experimentation conducted so that the effects of the cavitation jet can be fully understood. In some of the experimentation conducted in this analysis, critical heat flux was never achieved because of hardware limitations, so the full capacity of the jet was not determined. Since the entire purpose of this analysis was to delay the onset of critical heat flux, it is essential to know by how much it is delayed or if critical heat flux has been bypassed entirely.
REFERENCES


104


105


106


