Summer 2006

Micron-Level Actuator for Thermal-Fluid Control in Microchannels

Nurhak Erbas
Old Dominion University

Follow this and additional works at: https://digitalcommons.odu.edu/mae_etds
Part of the Mechanical Engineering Commons, Navigation, Guidance, Control and Dynamics Commons, and the Structures and Materials Commons

Recommended Citation
Erbas, Nurhak. "Micron-Level Actuator for Thermal-Fluid Control in Microchannels" (2006). Doctor of Philosophy (PhD), dissertation, Aerospace Engineering, Old Dominion University, DOI: 10.25777/zbhg-6z71
https://digitalcommons.odu.edu/mae_etds/54

This Dissertation is brought to you for free and open access by the Mechanical & Aerospace Engineering at ODU Digital Commons. It has been accepted for inclusion in Mechanical & Aerospace Engineering Theses & Dissertations by an authorized administrator of ODU Digital Commons. For more information, please contact digitalcommons@odu.edu.
MICRON-LEVEL ACTUATOR FOR THERMAL-FLUID CONTROL
IN MICROCHANNELS

by

Nurhak Erbas
M.Sc. December 2001, Istanbul Technical University, Istanbul, Turkey

A Dissertation Submitted to the Faculty of
Old Dominion University in Partial Fulfillment of the
Requirement for the Degree of

DOCTOR OF PHILOSOPHY
AEROSPACE ENGINEERING
OLD DOMINION UNIVERSITY
August 2006

Approved by:

Oktay Baysal (Director)

Osama Kandil (Member)

Gene Hou (Member)
Dedicated to my parents
Effectiveness of an actuator is investigated for thermal-flow control in microchannels. First, simulations of a single actuator in a quiescent external medium are performed in order to study the parameters characterizing the synthetic jet flow from the actuator. For this purpose, a simplified, two-dimensional configuration is considered. The membrane motion is modeled in a realistic manner as a moving boundary in order to accurately compute the flow inside the actuator cavity. The geometric and actuation parameters of the actuator are investigated to define the effectiveness of the jet flow. The study is done initially at macro scales. Then, the flow in the Knudsen number range of less than 0.1 is modeled starting with a conventional compressible Navier-Stokes solver valid for continuum approach. Its boundary conditions, however, are modified to account for the slip velocity and the temperature jump boundary conditions encountered in micron-level devices. Compressibility effects are also taken into account and modeled through the compressible flow solver. The utility of synthetic jet actuators for manipulating fluid flows has been shown for mostly macro- and mini-scale applications. To the best of the author’s knowledge, there have been only a few studies on micro-sized synthetic jets; also they have only been modeled assuming continuum flow regime with no-slip at the walls. Therefore, several issues must still be addressed for micron-scale
synthetic jets and also their applications to micron-level problems. Thus, as the second part of the study, a micron-level synthetic jet is proposed as a flow control device to manipulate the separated flow past a backward facing step in a microchannel. First, an uncontrolled flow past a backward facing step in a channel is computed. Then, a synthetic jet actuator is placed downstream of the step where the separation occurs. A large number of test cases have been analyzed. It is observed that the size of the separation bubble and its enstrophy are functions of the geometry of the actuator cavity and the membrane oscillation parameters. Considerable reduction in separation bubble size as well as in enstrophy is achieved using the actuator. Finally, a design for thermal management of a semiconductor device using the present actuator is introduced. For this purpose, a single microchip dissipating heat is placed in a two dimensional rectangular channel. Then, the different cavity and actuation parameters are considered in order to infer some characteristics of the effect of controlled synthetic jet thermal management. Using the actuator, a circulation region is generated on the top surface of the microelectronic chip. It is found that the fluctuating jet interacts with the channel flow and increases the convection rate by transferring linear momentum to the channel flow.

It is seen from the results of the computations that the synthetic jets can be utilized effectively to control separation in internal flow applications and that they guarantee an efficient thermal management of microelectronic devices. Therefore, the synthetic jet actuator proves itself to be an effective device for thermal-fluid control applications where low-speed flows are encountered.
ACKNOWLEDGMENTS

First, I thank my advisor Dr. Oktay Baysal, for his continuous help and support in the Ph.D. program and for being a great mentor during my time at ODU. He taught me how to write academic papers, how to work hard and most importantly had confidence in me when I needed it. I also wish to thank my dissertation committee members, Dr. Osama Kandil for his encouragement and Dr. Gene Hou for guiding me at various stages of getting my degree. I also would like to thank Mehti Koklu for his help in the projects we worked together.

I am grateful to Dr. Rustem Aslan, my teacher in the past and my Master’s advisor for getting me interested in Computational Fluid Dynamics and coming to United States. He inspired me the most as a great researcher and supported me to take this big step forward in my career.

I am also very much indebted to my great friends, Aysegul Gungor, Baris Ali Sen and Suna Cinar, for listening to me whenever I needed, for cheering me up at the most difficult times and for wonderful times away from the research.

Last, but not least, I thank my family: My parents, Mustafa Erbas and Hatun Erbas, for giving me life in the first place, for unconditional love and support and for encouragement to pursue my interests. My sister, Urun Erbas, for always being there for me, for listening to my complaints and frustrations and for believing in me. I could not do it without her support. My brother, Deniz Erbas, for his constant support. He helped me understand the path of my life.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF TABLES</td>
<td>vii</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>viii</td>
</tr>
<tr>
<td>NOMENCLATURE</td>
<td>xvii</td>
</tr>
<tr>
<td>CHAPTER I. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1. Motivation</td>
<td>1</td>
</tr>
<tr>
<td>1.2. Synopsis of Synthetic Jet Actuators</td>
<td>3</td>
</tr>
<tr>
<td>1.3. Literature Survey</td>
<td>5</td>
</tr>
<tr>
<td>1.3.1. Studies on Synthetic Jet Formation and Evolution</td>
<td>6</td>
</tr>
<tr>
<td>1.3.1.1. Computational Studies of Synthetic Jets</td>
<td>6</td>
</tr>
<tr>
<td>1.3.1.2. Experimental Studies of Synthetic Jets</td>
<td>14</td>
</tr>
<tr>
<td>1.3.2. Synthetic Jets to Control Separation Downstream of a Step</td>
<td>16</td>
</tr>
<tr>
<td>1.3.3. Synthetic Jets for Thermal Management</td>
<td>19</td>
</tr>
<tr>
<td>1.3.3. Other Applications of Synthetic Jets</td>
<td>28</td>
</tr>
<tr>
<td>CHAPTER II. PHYSICS of SLIP FLOW REGIME</td>
<td>32</td>
</tr>
<tr>
<td>2.1. MEMS and Microfluidics</td>
<td>32</td>
</tr>
<tr>
<td>2.2. Flow Modeling</td>
<td>35</td>
</tr>
<tr>
<td>2.2.1. Intermolecular Forces</td>
<td>35</td>
</tr>
<tr>
<td>2.2.2. Continuum Approach</td>
<td>38</td>
</tr>
<tr>
<td>2.2.3. Kinetic Gas Theory</td>
<td>40</td>
</tr>
<tr>
<td>2.2.4. Flow Regimes in Microfluidics</td>
<td>42</td>
</tr>
<tr>
<td>2.2.4.1. Gas Micro Flows</td>
<td>43</td>
</tr>
<tr>
<td>2.2.4.2. Liquid Micro Flows</td>
<td>47</td>
</tr>
<tr>
<td>2.3. Slip Flow Regime</td>
<td>49</td>
</tr>
<tr>
<td>2.3.1. Governing Equations</td>
<td>50</td>
</tr>
<tr>
<td>2.3.2. Boundary Conditions</td>
<td>53</td>
</tr>
<tr>
<td>CHAPTER III. MATHEMATICAL MODEL</td>
<td>56</td>
</tr>
<tr>
<td>3.1. Numerical Method</td>
<td>56</td>
</tr>
<tr>
<td>3.1.1. Governing Equations</td>
<td>57</td>
</tr>
<tr>
<td>3.1.2. Numerical Algorithm</td>
<td>60</td>
</tr>
<tr>
<td>3.1.2.1. Roe’s Flux Difference Spliting Technique</td>
<td>61</td>
</tr>
<tr>
<td>3.1.2.2. Moving Deforming Mesh</td>
<td>63</td>
</tr>
<tr>
<td>3.1.2. Initial and Boundary Conditions</td>
<td>65</td>
</tr>
<tr>
<td>3.2. Verification of Modified Boundary Conditions</td>
<td>69</td>
</tr>
<tr>
<td>3.2.1. Straight Micro Channel</td>
<td>70</td>
</tr>
<tr>
<td>3.2.1. Micro Filter</td>
<td>73</td>
</tr>
<tr>
<td>3.2.1. Backward Facing Step</td>
<td>78</td>
</tr>
</tbody>
</table>

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
CHAPTER IV. RESULTS and DISCUSSION .................................................................81
  4.1. Modeling and Characterization of Synthetic Jets ......................................81
    4.1.1. Computational Aspects .................................................................85
  4.1.2. Synthetic Jet Modeling .................................................................88
  4.1.3. Synthetic Jet Design Considerations ..............................................94
    4.1.3.1. Cavity Shape .................................................................95
    4.1.3.2. Throat Height .............................................................98
    4.1.3.3. Throat Width .............................................................104
    4.1.3.4. Membrane Oscillation Amplitude .......................................106
    4.1.3.5. Membrane Oscillation Frequency .......................................109
  4.1.4. Micron Level Synthetic Jets .........................................................112
  4.1.5. Effect of Ambient Temperature on Synthetic Jet Flow .....................116
  4.2. Control of Separated Flow past a Backward Facing Step in a Microchannel ........................................................................................................120
    4.2.1. Computational Aspects .........................................................122
    4.2.2. Computational Cases and Discussion .........................................124
  4.3. Thermal Management of Microelectronic Devices Using Synthetic Jets ........................................................................................................138
    4.3.1. Computational Aspects .............................................................140
    4.3.2. Computational Cases and Discussion ...........................................143
      4.3.2.1. Single Synthetic Jet Cooling ...........................................148
        4.3.2.1.1. Effect of Jet Location ...........................................148
        4.3.2.1.2. Effect of Throat Geometry .........................................151
        4.3.2.1.3. Jet Cooling for Various Heat Fluxes .............................153
      4.3.2.2. Multiple Synthetic Jet Cooling ........................................156
        4.3.2.2.1. Cooling with Two Jets ........................................156
          4.3.2.2.1.1. Effect of Jet Locations ........................................156
          4.3.2.2.1.2. Effect of Phase Angle ...........................................159
        4.3.2.2.2. Cooling with Three Jets ......................................162

CHAPTER V. CONCLUSIONS .....................................................................................166

REFERENCES .............................................................................................................175

CURRICULUM VITA .....................................................................................................181

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
LIST OF TABLES

Table | Page
--- | ---
3.1 | Case definitions for the micro filter.| 75
5.1. | Computational cases and synthetic jet parameters for quiescent external flow case; R: rectangular, T: triangular. | 100
5.2. | Mean jet velocities at the maximum expulsion stage for Cases 2-8. | 108
5.3. | Mean jet velocities at the maximum expulsion stage for Cases 2, 9-11. | 111
5.4. | Mean jet velocities at the maximum expulsion stage for Cases 1, 12-16. | 114
5.5. | Mean jet velocities at the maximum expulsion stage for Cases 2, 17-23. | 117
5.6. | Computational cases: Definitions and results. $d_o/h_o=1., W/d_o=20., H/d_o=4.$ | 118
5.7. | Computational cases: Definitions and results. $d_o/h_o=1., W/d_o=20., H/d_o=4.$ | 122
5.8. | Description of computational cases: $h_c=12.5 \mu m.$ | 129
5.9. | Results of computational cases: $h_c=12.5 \mu m.$ | 130
5.10. | Case 1: No- synthetic jet cooling. | 144
5.11. | Definitions of computational cases for single synthetic jet control, $f=10$ kH z. | 147
5.12. | No-control cases for various heat fluxes. | 147
5.13. | Definitions of computational cases for multiple synthetic jet control, $q^*=12 \text{ W/m}^2, h_o/d_o=0.$ | 147
5.14. | Results of computational cases of synthetic jet cooling. | 148

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1. Principle of synthetic jets</td>
<td>3</td>
</tr>
<tr>
<td>1.2. Different synthetic jet configurations</td>
<td>4</td>
</tr>
<tr>
<td>1.3. Synthetic jet configuration with triangular-shaped cavity</td>
<td>5</td>
</tr>
<tr>
<td>1.4. Dimensions of the backward facing setup (Rediniotis et al [31])</td>
<td>18</td>
</tr>
<tr>
<td>1.5. Major causes of electronic failures (Yeh and Chu [50])</td>
<td>20</td>
</tr>
<tr>
<td>1.6. ITRS roadmap(s) and CPU historical data for high-performance computers (Prasher et al [55])</td>
<td>21</td>
</tr>
<tr>
<td>1.7. Microjet orifice plate designs for laptop cooling; a) square, single-orifice plate b) cut, multi-orifice plate (Campbell et al [36])</td>
<td>23</td>
</tr>
<tr>
<td>1.8. Basic principle of operation of a synthetic jet ejector heat sink (Mahalingam and Glezer [37])</td>
<td>24</td>
</tr>
<tr>
<td>1.9. A cross-sectional view of the prototype near the middle of one jet orifice (Beratlis and Smith [55])</td>
<td>25</td>
</tr>
<tr>
<td>1.10. Schematic diagram of a synthetic jet ejector heat sink module a) side view b) top view (Mahalingam et al. [56])</td>
<td>26</td>
</tr>
<tr>
<td>2.1. Characteristic lengths of typical fluidic microsystems, with the range of Knudsen number corresponding to standard conditions (Karniadakis and Beskok [70])</td>
<td>45</td>
</tr>
<tr>
<td>2.2. Molecular and continuum flow models (Gad-el-Hak [3])</td>
<td>46</td>
</tr>
<tr>
<td>3.1. Cell-center type boundary conditions</td>
<td>66</td>
</tr>
<tr>
<td>3.2. Cell-face type boundary conditions</td>
<td>66</td>
</tr>
</tbody>
</table>
Figure Page

3.3. Computational domain for straight channel: every 3rd grid point is shown...........71

3.4. Deviation of centerline pressure (Eq. 3.37) distribution through the micro duct
for different computational models, $Kn_{in} = 0.088$.................................72

3.5. Variation of slip velocity along the microduct wall, $Kn_{in} = 0.088$.............73

3.6. Cross-sectional view and the characteristics dimensions of the micro filter........74

3.7. Micro filter and its computational domain.....................................................74

3.8. a) Normalized streamwise velocity variations along the centerline of micro filter,
b) normalized streamwise temperature variations along the centerline of micro
filter...............................................................................................................76

3.9. Variation of shear stress along micro filter wall. Case I (left) and
Case II (right)...............................................................................................77

3.10. Separated flow in a micro backward facing step.........................................78

3.11. Micro backward facing step with its numerical grid.....................................78

3.12. Streamwise variation of pressure..................................................................79

3.13. Variation of normalized streamwise component of velocity.....................79

4.1. Synthetic jet flow configuration.................................................................81

4.2. Position and shape of membrane at its maximum deflection.....................82

4.3. Separated flow in a channel with backward facing step............................85

4.4. Synthetic jet actuator in a microchannel with backward facing step...........85

4.5. Single-chip package geometry (Yeh and Chu [50]).....................................88

4.6. Schematic of the simplified microchip geometry in two-dimensional space.....88
<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1.</td>
<td>91</td>
</tr>
<tr>
<td>5.2.</td>
<td>92</td>
</tr>
<tr>
<td>5.3.</td>
<td>93</td>
</tr>
<tr>
<td>5.4.</td>
<td>95</td>
</tr>
<tr>
<td>5.5.</td>
<td>96</td>
</tr>
<tr>
<td>5.6.</td>
<td>97</td>
</tr>
<tr>
<td>5.7.</td>
<td>98</td>
</tr>
<tr>
<td>5.8.</td>
<td>99</td>
</tr>
<tr>
<td>5.9.</td>
<td>99</td>
</tr>
<tr>
<td>5.10.</td>
<td>101</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
</tr>
<tr>
<td>5.11. Case 2: Velocity profiles at the throat exit at four stages of one membrane oscillation cycle; a) u-velocities, b) v-velocities.</td>
<td>102</td>
</tr>
<tr>
<td>5.12. Comparison of velocity profiles at the four stages of membrane oscillation cycle: a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.</td>
<td>103</td>
</tr>
<tr>
<td>5.13. Throat geometries: a) Case 3; b) Cases 4-8.</td>
<td>104</td>
</tr>
<tr>
<td>5.14. Cases 3-8: Plot of vorticity contours at the four stages of membrane oscillation cycle: a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.</td>
<td>105</td>
</tr>
<tr>
<td>5.15. Spanwise velocity profiles for Cases 2-8; a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.</td>
<td>106</td>
</tr>
<tr>
<td>5.16. Streamwise velocity profiles for Cases 2-8; a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.</td>
<td>107</td>
</tr>
<tr>
<td>5.17. Mean streamwise velocity at the orifice exit vs. orifice height.</td>
<td>108</td>
</tr>
<tr>
<td>5.18. Case 8: Velocity profiles at five locations: at maximum expulsion stage.</td>
<td>109</td>
</tr>
<tr>
<td>5.19. Plot of vorticity contours for different $d_o/d_{ref}$ ratios: a) Case 9: 0.2, b) Case 2: 1.0, c) Case 10: 2.0, d) Case 11: 5.0.</td>
<td>110</td>
</tr>
<tr>
<td>5.20. Mean streamwise velocity at the orifice exit vs. orifice width.</td>
<td>111</td>
</tr>
<tr>
<td>5.21. Plot of vorticity contours for different $A/d_{ref}$ ratios: a) 0.05, b) 0.1, c) 0.2, d) 0.4, e) 0.6, f) 0.8.</td>
<td>112</td>
</tr>
<tr>
<td>5.22. Cases 1, 12-16: Velocity profiles at the throat exit; a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.</td>
<td>113</td>
</tr>
</tbody>
</table>
Figure | Page
--- | ---
5.23. Mean streamwise velocity at the orifice exit vs. membrane oscillation amplitude | 114
5.24. Plot of vorticity contours for different frequencies: a) $f = 250$ Hz, b) $f = 500$ Hz, c) $f = 750$ Hz d) $f = 1000$ Hz, e) $f = 1250$ Hz, f) $f = 1500$ Hz, g) $f = 2000$ Hz | 115
5.25. Velocity profiles for Cases 2, 17-23: a) maximum expulsion, b) minimum volume, c) maximum ingestion, d) maximum volume | 116
5.26. Mean streamwise velocity at the orifice exit vs. membrane oscillation frequency | 117
5.27. Plot of vorticity contours: a) Case 1, b) Case 2, c) Case 3, d) Case 4, e) Case 5, f) Case 6, g) Case 7 | 119
5.28. $u$-velocity profiles for Cases 1-7; a) maximum expulsion; b) minimum volume c) maximum ingestion; d) maximum volume | 120
5.29. $v$-velocity profiles for Cases 1-7; a) maximum expulsion; b) minimum volume c) maximum ingestion; d) maximum volume | 121
5.30. Plot of vorticity contours: a) Case 1, b) Case 2, c) Case 3, d) Case 4, e) Case 5, f) Case 6 | 123
5.31. $u$-velocity profiles for Cases 1-3 at maximum expulsion stage | 124
5.32. $v$-velocity profiles for Cases 4-6 at maximum expulsion stage | 124
5.33. Uncontrolled Case: a) Computational domain and boundary conditions, b) numerical grid | 126
5.34. A controlled Case: a) Computational domain and boundary conditions, b) numerical grid | 127
5.35. Plot of vorticity contours at four stages of one membrane cycle, \( M=0.00001 \);
a) maximum expulsion; b) minimum volume, c) maximum ingestion,
d) maximum volume ................................................................. 128

5.36. The velocity profile at the orifice exit at four different stages of the cycle,
a) \( u \)-velocity, b) \( v \)-velocity ................................................................. 128

5.37. Reference case: Separated flow without control in a channel with backward facing step..................................................... 131

5.38. Flow past a micro backward facing step with flow control illustrated via stream traces. Close-up of separation region; a) Case1, b) Case2, c) Case3,
d) Case4 ............................................................................................. 132

5.39. Case 1-4: The velocity profile at the orifice exit at four different stages of the cycle; a) maximum expulsion, b) minimum volume, c) maximum ingestion,
d) maximum volume ........................................................................ 133

5.40. Case 8: \( d_o=0.2 h_c \). Flow control illustrated via stream traces and vorticity contours. Close-up of separation region; a) maximum expulsion,
b) minimum volume, c) maximum ingestion, d) maximum volume ....... 135

5.41. Case 8: \( d_o=0.2 h_c \). The velocity profile at the orifice exit at four different stages of the cycle; a) \( u \)-velocity, b) \( v \)-velocity ..................................................... 135

5.42. Velocity vectors: a) Reference case, b) Case 8 at maximum expulsion stage,
c) Case 8 at minimum volume stage, d) Case 8 at maximum ingestion stage,
d) Case 8 at maximum volume stage .................................................. 136

5.43. Case 1, 7-9: Effect of throat width on separation control ....................... 137
<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.44. Case 1, 5-6: Effect of throat height on separation control</td>
<td>138</td>
</tr>
<tr>
<td>5.45. Case 1, 10-11: Effect of cavity height on separation control</td>
<td>138</td>
</tr>
<tr>
<td>5.46. Case 1, 12-13: Effect of cavity width on separation control</td>
<td>139</td>
</tr>
<tr>
<td>5.47. Case 1, 14-16: Effect of membrane oscillation amplitude on separation control</td>
<td>139</td>
</tr>
<tr>
<td>5.48. Case 1, 17, 18: Effect of membrane oscillation frequency on separation control</td>
<td>140</td>
</tr>
<tr>
<td>5.49. Computational domain and boundary conditions for no synthetic jet cooling, (every fourth grid line shown for clarity)</td>
<td>142</td>
</tr>
<tr>
<td>5.50. Computational domain and boundary conditions for a synthetic jet cooling configuration, (every fourth grid line shown for clarity)</td>
<td>143</td>
</tr>
<tr>
<td>5.51. Case 1: Flow and temperature fields; top: no-slip boundary condition, bottom: slip boundary condition</td>
<td>144</td>
</tr>
<tr>
<td>5.52. Case 1: Comparison of temperature profiles over the top surface of microchip for slip and no-slip boundary conditions</td>
<td>145</td>
</tr>
<tr>
<td>5.53. Selected synthetic jet actuator geometry to test effect of jet location and its numerical grid</td>
<td>149</td>
</tr>
<tr>
<td>5.54. Five stations to test the effect of the location</td>
<td>150</td>
</tr>
<tr>
<td>5.55. Instantaneous vorticity contour plots demonstrating the circulation region. Top: Case 2, bottom: Case 3</td>
<td>150</td>
</tr>
<tr>
<td>5.56. Cases 2-6: Instantaneous temperature fields</td>
<td>151</td>
</tr>
<tr>
<td>5.57. Temperature plots over the microchip surface; Cases 1-6</td>
<td>152</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
</tr>
<tr>
<td>5.58.</td>
<td>Cases 3, 7-11: Different throat geometries with a close-up of temperature field.</td>
</tr>
<tr>
<td>5.59.</td>
<td>Convergence history of Case 11: Heat transfer coefficient per unit area versus membrane cycle number.</td>
</tr>
<tr>
<td>5.60.</td>
<td><em>Left:</em> Mean temperature over the microchip surface versus heat flux; <em>right:</em> heat transfer coefficient per unit area versus heat flux.</td>
</tr>
<tr>
<td>5.61.</td>
<td>Cases 12-16: Instantaneous temperature fields at maximum expulsion stage.</td>
</tr>
<tr>
<td>5.62.</td>
<td>Cases 1, 12-16: Percent increase in heat transfer coefficient.</td>
</tr>
<tr>
<td>5.63.</td>
<td>Four configurations of synthetic jets; a) Case 17, b) Case 18, c) Case 19, d) Case 20.</td>
</tr>
<tr>
<td>5.64.</td>
<td>Instantaneous temperature fields via temperature contours; a) Case 17, b) Case 18, c) Case 19, d) Case 20.</td>
</tr>
<tr>
<td>5.65.</td>
<td>Temperature plots over the microchip surface; Cases 1, 17-20.</td>
</tr>
<tr>
<td>5.66.</td>
<td>Cases 21-23: Instantaneous temperature fields illustrated via temperature contours.</td>
</tr>
<tr>
<td>5.67.</td>
<td>Temperature plots over the microchip surface; Cases 1, 18, 21-23.</td>
</tr>
<tr>
<td>5.68.</td>
<td>Case 21: Instantaneous vorticity contours demonstrating the circulation region.</td>
</tr>
<tr>
<td>5.69.</td>
<td>Case 21: Heat transfer coefficient per unit area versus membrane cycle number.</td>
</tr>
<tr>
<td>5.70.</td>
<td>Cases 24-27: Cooling configuration with three synthetic jets.</td>
</tr>
</tbody>
</table>
Figure 5.71. Instantaneous temperature fields illustrated via temperature contours.

a) Case 24, b) Case 25, c) Case 26, d) Case 27 .................................165

Figure 5.72. Instantaneous vorticity contours demonstrating the circulation region.

a) Case 24, b) Case 25, c) Case 26, d) Case 27 .................................166
NOMENCLATURE

Constants and Parameters

\( c_h \) Heat transfer coefficient
\( c_{ij}, d_{ij} \) Parameters particular to the pair of interacting molecules
\( k_B \) Boltzmann’s constant
\( n \) Number density
\( R \) Universal gas constant
\( \overline{R} \) Specific gas constant
\( \varepsilon \) Energy scale in Lennard-Jones Potential
\( \gamma \) Ideal gas constant
\( \zeta \) Second coefficient of viscosity
\( \sigma_v \) Momentum accommodation coefficients
\( \sigma_t \) Thermal accommodation coefficients

Non-Dimensional Numbers

\( Ec \) Eckert number
\( Kn \) Knudsen number
\( k_r \) Reduced frequency
\( Ma \) Mach number
\( L_o \) Stroke length
\( Pr \) Prandtl number
\( Re \) Reynolds number
\( St \) Stokes number
\( Str \) Strouhal number

Time and Length Scales

\( A \) Maximum membrane oscillation amplitude
\( A_c \) Cross sectional area of the channel
\( d \) Molecular diameter
\( d_o \) Throat width
\( \delta \) Mean molecular spacing
\( f \) Membrane forcing frequency
\( h_c \) Outlet channel height
\( h_t \) Throat height
\( H \) Cavity height
\( L \) Channel length, characteristic dimension
\( \lambda \)  
Mean free path

\( P \)  
Wetted perimeter of the channel

\( r \)  
Distance separating the molecules \( i \) and \( j \)

\( s \)  
Step height

\( \sigma \)  
Length parameter in Lennard-Jones Potential

\( \Omega_s \)  
Area of the separated region

\( \tau \)  
Period of oscillation in Lennard-Jones Potential

\( \omega \)  
Membrane oscillation frequency

\( W \)  
Cavity width

\( x_{jet} \)  
Placement of synthetic jet

**Numerical Parameters**

\( \Omega \)  
Control volume, area of computational domain

\( \partial \Omega \)  
Control surface

\( \hat{n} \)  
Outward normal unit vector

\( \hat{s} \)  
Tangential unit vector

\( x, y \)  
Cartesian coordinate system

\( \mathbb{I} \)  
Unit tensor

**Differential Operators**

\( \nabla^2 \)  
Laplacian

\( \nabla \cdot \)  
Divergence

\( \nabla \times \)  
Curl

**Fluid and Thermal Parameters**

\( a \)  
Speed of sound

\( \bar{c} \)  
Mean-square molecular speed

\( C_p, C_v \)  
Specific heats

\( e, E \)  
Internal, total specific energy

\( \tilde{f} \)  
External forces acting on the control volume

\( f_o \)  
Maxwellian distribution function

\( F_{ij} \)  
Force between two interacting molecules

\( J_{ens} \)  
Enstrophy

\( k \)  
Thermal conductivity

\( M \)  
Molar mass of the gas

\( \nu, \mu \)  
Kinematic, dynamic viscosities

\( p \)  
Pressure

\( \bar{q} \)  
Heat flux vector

\( \bar{q}^\prime \)  
Heat flux per unit area

---

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
$\rho$  Density
$\sigma$  Normal stress tensor
$\tau$  Viscous stress tensor
$T$  Temperature
$U$  Velocity
$\vec{v}$  Velocity field
$V_{ij}$  Lennard-Jones potential
$V_{jet}$  Fluctuating jet velocity
CHAPTER I

INTRODUCTION

1.1. Motivation

Macro and mini synthetic jets are popular actuators for external flow applications and have been proposed as versatile flow control devices for micro-electro-mechanical systems (MEMS) applications (Aslan et al. [1]). As new applications of MEMS arise, it has been acknowledged that the operation of miniaturized devices often challenges the validity of the conventional methods developed for modeling of macroscopic devices (Karniadakis and Beskok [2]; Gad-el-Hak [3]; Liou and Fang [4]; Nguyen and Werely [5]). In the last two decades, despite the diverse prospects and fast growth of MEMS, thrive for further miniaturization of device scales is still faced with the challenge of better understanding of micron- and sub-micron-scale physics (Gad-el-Hak [3]). The primary concerns are the implementation of micro-scale-specific physical models and treatment of the complex geometry.

There have been numerous studies done on synthetic jet actuators both computationally (Kral et al. [6]; Mallinson et al. [7]; Mittal et al. [8]) and experimentally (Smith and Glezer [9]; Gillarranz et al. [10]). The utility of synthetic jet actuators for manipulating fluid flows has been shown for mostly macro- and mini-scale applications. To the best of the author’s knowledge, there have been only a few studies on micro-sized synthetic jets; also they have only been modeled assuming continuum flow regime with no-slip at the walls (Mallinson et al. [11]; Timchenko et al. [12]; Lockerby and Carpenter...
Therefore, several issues must still be addressed for micron-scale synthetic jets and also their applications to micron-level problems.

Major differences in the present study are twofold. First, the present focus is on microflows where the flow physics is different. Fluid flows in small devices differ from those in macroscopic devices, and understanding the unfamiliar physics of fluids involved in microflows is a key issue for microfluidics. A key non-dimensional parameter for gas micro flows is Knudsen (Kn) number. Depending on the Kn number range, a full continuum or a full free-molecular analysis may be applicable (Kamiadakis and Beskok [2]; Gad-el-Hak [3]; Beskok \textit{et al} [14]; Ho and Tai [15]). If Kn number is smaller than 0.1, the traditional Navier-Stokes (NS) solvers with boundary condition modifications can be employed effectively (Arkilic [16]; Beskok [17]; Zohar \textit{et al} [18]; Baysal and Aslan [19]). Second, although studies have shown improved results of synthetic jet applications, for instance, to flow separation downstream of a step and cooling of electronic devices, our understanding of what affects the performance of the synthetic jet is rather incomplete. Therefore, a thorough study needs to be done to investigate the effect of geometrical dimensions and actuation parameters on synthetic jet performance for thermal-fluid control applications and this forms the main objective of the current study. The present study includes a high-fidelity modeling of the geometry to include both the cavity and the motion of the piezoelectric membrane.

Design process of synthetic jets is usually time consuming and dependent on trial-and-error tests. Also, most experimental studies can not predict the flow phenomena occurring in the actuator cavity. Computational Fluid Dynamics (CFD) can play a crucial role in modeling of the flow configuration in an accurate and cost-effective manner.
Therefore, numerical simulation is used in this study for modeling of synthetic jet actuators with a jet into a quiescent flow as well as for applications of micro synthetic jets to separation control and thermal management of microelectronic devices.

1.2. Synopsis of Synthetic Jet Actuators

A synthetic jet is a zero-net-mass-flow fluidic device (Aslan et al [1]; Smith and Glezer [20]; Glezer and Amitay [21]), which is formed by the working fluid in the flow system in which it is embedded. As such, it eliminates the need for additional hardware such as fluid ducting (Kral et al [6]; Smith and Glezer [9]). Even though the jet is formed without any mass injection to the system, the net effect over one cycle of the membrane motion is a finite streamwise momentum.

In a synthetic jet, the actuating flow is generated at the orifice of a cavity by oscillating a membrane opposite to the orifice (Fig. 1.1). A pair of vortices forms at the orifice during the expulsion phase of the membrane oscillation cycle of the synthetic jet actuator. Fluctuating jet flow then interacts with the main flow and transfers linear momentum to the main flow.

![Figure 1.1. Principle of a synthetic jet.](image-url)
The oscillating the membrane of the device is driven either by a piezoelectric or electrostatic or electromagnetic actuator or by other means at its resonance frequencies (Mallinson et al [7]). Thus, the electrical power input to drive the actuator is small (Baysal et al [22]). When the membrane is forced to oscillate, the fluid inside the cavity is expelled through the orifice as membrane moves upward creating a vortex pair (a vortex ring for an axisymmetric flow case), which moves outward under its own momentum (Aslan et al [19]; Edis et al [23]). As the membrane moves downward the same amount of fluid is entrained from the external flow into the cavity. If the vortex structure can shed from the orifice by its own induced velocity, then it will not be affected by the entrainment of the fluid into the cavity. Thus, the total effect of the synthetic jet will be an increase of the momentum transferred to the external flow with zero net mass flux.

There are a number of potential geometries that can be employed for synthetic jets. Some of these presented first in Kiddy et al [24] are shown in Figure 1.2a. Shown in Figure 1.2b are the synthetic jet actuators that have been implemented with different membrane configurations Utturkar et al [25]. Also, different types of jet exit shapes are possible, such as, circular and rectangular. In our study, we focused on two-dimensional

![Figure 1.2. Different synthetic jet configurations](image-url)

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
actuator configurations with mainly two different actuator cavity geometries: rectangular-shaped and triangular-shaped cavity. One of these configurations is shown in Figure 1.3.

![Diagram of synthetic jet configuration with triangular-shaped cavity](https://via.placeholder.com/150)

**Figure 1.3.** Synthetic jet configuration with triangular-shaped cavity.

1.3. Literature Survey

The studies on synthetic jet actuators have been performed experimentally or computationally in order to better understand their flow characteristics. Some of these studies have attempted to define a criterion for jet formation of synthetic jets (Smith and Swift [26]; Utturkar *et al.* [27]; Holman *et al.* [28]).

Synthetic jets have been used as active control devices in areas ranging from jet vectoring (Smith and Glezer [29]), to separation control of both external and internal flows (Mittal *et al.* [8]; Baysal *et al.* [22]; McCormick [30]; Rediniotis *et al.* [31]; Jain *et al.* [32]), to enhancing mixing (Mautner [33], Smith and Glezer [20]), and to improving heat transfer (Erbas *et al.* [34]; Timchenko *et al.* [35]; Campbell *et al.* [36]; Mahalingam and
Glezer [37]). Despite the considerable advance in control theory, applications of flow control still remain as a challenging area.

In this thesis, synthetic jets are used for controlling the flow in a microchannel with a backward facing step and for enhancing heat transfer from a microelectronic device. Therefore, the following sections talk about previous studies that have been done on some characteristics of synthetic jets and their applications to thermal-fluid control.

1.3.1. Studies on Synthetic Jet Formation and Evolution

1.3.1.1. Computational Studies of Synthetic Jets

Kral et al [6] performed incompressible, two dimensional simulations of a synthetic jet forming into a quiescent external flow. They investigated both laminar and turbulent synthetic jets. In their study, the flow within the cavity was not calculated and an analytic velocity profile was assumed at the orifice exit to simulate an isolated synthetic jet using the RANS approach in the form of;

\[ \bar{u}_n(\xi = 0,\eta,t) = U_0 f(\eta) \sin(\omega t) \]  

(1.1)

where \( \xi \) denoted the streamwise direction, \( \eta \) denoted the cross-stream direction and \( \bar{u}_n \) was the streamwise component of velocity. \( f(\eta) \) was the function of the spatial velocity defined as follows:

\[ f(\eta) = \begin{cases} 1 \\ \sin(\pi \eta) \\ \sin^2(\pi \eta) \end{cases} \]  

(1.2)
The modeled actuator mimics a suction/blowing boundary condition at the orifice and creates a symmetric velocity profile at suction and blowing phases. It will be shown later in the present study that synthetic jet displays different profile characteristics.

In their study, they initially performed laminar simulations. They demonstrated that the laminar jet developed a train of vortex pairs but did not capture the breakdown of the vortices. Then, they incorporated a turbulence model in the simulation to artificially increase viscous diffusion and mimic breakup of the vortex train. They compared steady jets, pulsed jets and synthetic jets acquiring the same momentum at the orifice exit. Good agreement was obtained with measured mean jet velocity profiles.

Although they were not able to capture the flow characteristics inside the cavity and their effect on the jet velocity, their study suggested that the modeled boundary condition could capture some characteristics of the synthetic jet.

A similar study in the sense of modeling the actuator was performed by Mallinson et al [7]. They studied the synthetic jet flow generated by a circular orifice both experimentally and computationally. In the experimental study, they measured the jet velocity using single-component hot wire anemometry. For their computational study, they modeled the actuator as a boundary condition for the velocity normal to the orifice opening:

\[ u_s(t) = u_o \sin(\omega t) \]  

(1.3)

They assumed a top-hat function for the velocity distribution following the conclusions of Kral et al [6] that this form of distribution provided better predictions for planar synthetic jet actuators.
Initially, they investigated the synthetic jet for a specific geometry and membrane forcing frequency. They found some discrepancy between experimental and computational centerline distribution of the mean and fluctuating velocity. They attempted to explain this discrepancy by giving three reasons: lack of sufficient grid points, modeling the actuator with a boundary condition, or a possible turbulent dissipation which could only be captured using a 3-D model of the jet.

As the second part of the study, they examined the effect of varying the forcing frequency, orifice diameter and cavity height, while the orifice height and cavity diameter were held constant. They observed that two non-dimensional parameters based on orifice diameter defining the jet performance are Reynolds number and Strouhal number. As the Re number was increased, the St number initially decreased before increasing for Re>400. They also found that the maximum velocity for a particular actuator configuration would seem to depend on a balance between inertia (membrane forcing) and viscous (orifice boundary layer) forces.

Rizetta et al [38] investigated the flow field created by a synthetic-jet actuating device numerically using DNS to solve unsteady compressible Navier-Stokes equations for both two-dimensional and three-dimensional cases. Re number ranged from 750 to 1500 and the jet Mach number was 0.065. Unlike the previous studies by Kral et al (1997) and Mallinson et al (1998), they modeled both the interior of the actuator cavity and the external jet flow field. They used a decoupling technique for the calculations. The flow inside the cavity was simulated with an oscillating boundary condition prescribed at the lower boundary of the cavity. The velocity profile at the orifice exit was recorded for one complete cycle after several oscillatory cycles when the flow field
reached a periodic behavior. Then, this recorded profile was specified as a boundary condition for the exterior domain calculations. For their 3-D computations, they imposed symmetry planes at the jet centerline and at mid-span, thus only modeling the quarter of the whole computational domain in order to reduce the computational cost.

The results of the 2-D and 3-D simulations were compared with the experimental results. They found that the resultant jet velocity profiles as well as exterior flow fields of 2D simulations differed from commonly employed analytic models. The velocity profiles were periodic in time and it was possible to obtain the temporal statistics over several oscillation cycles, whereas 3D simulations showed a breakdown of the vortices close to the orifice exit. Also, they were able to capture the spanwise instability that led to a breakup of the coherent vortex structure with 3D simulations. This phenomenon could not be predicted in 2-D simulations.

Lee and Goldstein [39] studied the interaction of two dimensional synthetic jets using direct numerical simulation. They were able to model half of the physical domain by prescribing a periodic boundary condition. They investigated the effects of flow and geometric parameters on the resulting flow field.

They observed that jet formation to be highly sensitive to the jet Re number, whereas jet evolution was affected by Strouhal (Str) number. Re number dependence was examined by maintaining \( Str = 0.0628 \) while changing the Re number through variations of kinematic viscosity. Re number was found to have an effect of changing the velocity profile. \( Str \) number dependency was examined by fixing the Re number while varying \( Str \) number through changes in frequency. As they decreased the \( Str \) number, only a
slight tendency to increase the magnitude of spanwise and streamwise jet velocities was observed, while the vorticity remains mostly unchanged.

The shape of the lip and depth of the cavity were also found to be important parameters. They used three types of lip geometry to examine the effect of lip geometry. There were no substantial changes in the external or internal flows. However, an increase in the celerity of the ejected vortices was observed as the cavity becomes shallower. The half cavity showed a smaller circulation cell and the quarter cavity case does not even form a circulation cell.

Utturkar et al [25] examined the sensitivity of synthetic jets to the design of the jet cavity numerically by using an incompressible Navier-Stokes solver. They investigated the effect of varying the cavity aspect ratio as well as the placement of the oscillating membrane of the actuator cavity. They studied the synthetic jet for a quiescent external flow and also in the vicinity of an external flow. The results of their study showed that the details of the cavity and the placement of the membrane did not play a crucial role on the performance of the synthetic jet in terms of the vortex dynamics and the jet velocity profile. Integral parameters of the velocity profile showed less than a 7% deviation among all of the cavity designs. Therefore, they suggested these changes be considered as a requirement to satisfy design/deployment constraints. In their study, they did not take account of compressibility effect, but expected that a compressible flow inside the cavity would be more sensitive to the cavity design.

Utturkar et al [27] studied synthetic jets both experimentally and computationally and proposed a jet formation criterion for two dimensional as well as axisymmetric jets. The jet formation was defined as the mean outward velocity along the jet axis and
corresponded to the clear formation of shed vortices. They showed that the synthetic jet formation was governed by the Strouhal number which depends on the oscillation frequency of the membrane, the mean jet velocity and the width of the orifice for 2D jets (diameter of the orifice for circular jets).

They performed an order-of-magnitude analysis and defined their jet formation criterion as $\text{Re}/S^2 > K$. They suggested that $K$ was approximately 2 and 0.16 for 2-D and axisymmetric jets, respectively. Their experimental results were found to be in good agreement with the criterion. They compared their results with the available experimental and computational data in the literature. They suggested this criterion to be valid for relatively thick orifice plates with $w/d > 2$ in which the orifice flow was hypothesized to be nearly fully developed. Also, the approximation they made was only valid at low Stokes numbers where the velocity profile was parabolic.

Mallinson et al [11] studied two dimensional micro-fabricated synthetic jet actuators numerically employing a moving membrane boundary condition as it had been shown by other studies in the literature that it closely represented the physical phenomenon. They stated the lack of studies on micro-sized synthetic jet actuator flows. In their study, the Knudsen number based on the orifice diameter ($d=200 \ \mu m$) was $10^{-5}$ so that they modeled the flow using Navier-Stokes equations according to continuum hypothesis.

Their laminar simulations indicated that the vortex expelled from the cavity was seen to diffuse rapidly, which produced a flow pattern similar to that of turbulent flow. However, the primary vortex was observed to be relatively weaker than the secondary vortex. They explained this difference from the macro-scaled jet flows as the domination of the diffusion terms in Navier-Stokes equations due to smaller scales. The cavity and
orifice flows were found to behave similarly to equivalent macro-sized actuators. The actuator output was observed to vary linearly with the product of membrane displacement and frequency.

Timchenko et al [12] studied the effect of including the compressibility in the numerical simulations of synthetic jet actuators. They modeled the membrane as a sinusoidally moving boundary. The flow was simulated with $Kn<0.01$ so that the continuum approach using conventional conservation equations was valid. They performed both incompressible and compressible computations.

They compared numerical results obtained with incompressible and compressible computations for two different geometries with orifice diameters equal to 20 μm and 40 μm and the length of the orifice passage equal to 50 μm. They found significant differences in jet velocity between the incompressible and compressible solutions for the 20 μm orifice, whereas only a slight difference was observed for the 40 μm orifice. They showed that compressibility effects caused a phase lag between the orifice exit velocity and the motion of the membrane. Also, it was observed that the jet velocity at the orifice exit was highly dependent on the cavity volume. A further study revealed that a raise in the membrane amplitude increased the jet velocity, and increasing the membrane frequency did not necessarily result in an increase in the jet velocity. Finally, they concluded that the compressibility effects could not be neglected for modeling the flows at micro scales.

Fugal et al [40] studied two-dimensional synthetic jet formation using incompressible, Reynolds-averaged, Navier-Stokes equations. Turbulence effects were modeled using the standard $k$-$ε$ turbulence model with wall functions. They excluded the
cavity in their model prescribing an inlet boundary that consisted of a harmonic, top hat velocity profile as $u_n(t) = u_0 \sin(\omega t)$. The centerline of the jet was modeled with a symmetry boundary condition to reduce the computational time required to obtain a solution.

They produced results in terms of the power required to form the jet, the downstream momentum flux of the jet, and the location of the stagnation point. They investigated the synthetic jets for two exit geometries, a sharp exit and a rounded exit, and various stroke lengths. The power required to form the jet was shown to be significantly lower for the rounded exit than the sharp exit. They found that the jet formation threshold was higher for the rounded exit than the sharp exit. The effectiveness of the round exit is found to exceed that of sharp exit.

Gallas et al [41] studied synthetic jet actuators exhausting into a quiescent medium both numerically and experimentally. The geometry of the synthetic jet actuator was characterized by a sharp-edged orifice. They investigated both linear and non-linear losses of the flow field encountered in the orifice.

Ravi et al [42] studied the three-dimensional synthetic jet flow fields using direct numerical simulation. The motion of the membrane was modeled using appropriate boundary conditions at the bottom of the cavity. They focused on examining the formation of 3D synthetic jets for three different slot aspect ratios. The effect of the slot aspect ratio was realized by the jet velocity profiles and vortex dynamics.

Holman et al [28] performed both numerical simulations and experiments for synthetic jets with two-dimensional and axisymmetric orifices with a radius of curvature and proposed a jet formation criterion. They defined a "jet formation" as the appearance
of a time-averaged velocity directed outward along the jet axis. For the jet formation to be observed, they suggested, a subsequent convection of vortex pairs (or rings) must be present. Their jet formation criterion is dependent of some non-dimensional parameters, such as Reynolds number, Re, and Stokes number, $St$, as well as the orifice geometry and characterized as $K=\text{Re}/St^2$. They proposed that the jet would be formed when $K=1$ and $K=1.6$ for two-dimensional and axisymmetric orifices, respectively.

1.3.1.2. Experimental Studies of Synthetic Jets

Smith and Glezer [9] studied experimentally the formation and the evolution of synthetic jets in detail and demonstrated the interaction between two adjacent synthetic jets situated side by side and driven with different phase angles. In their experiment, the synthetic jet was formed in air at a rectangular orifice and the jet was synthesized by a membrane driven at resonance by a centrally bonded piezoceramic disk. Schlieren visualization was used to measure the velocity field of the synthetic jet.

In their study, they showed that synthetic jets exhibit a standing vortex near the orifice exit and the evolution of the synthetic jet near its exit plane was dominated by the time periodic formation. They explained this as being due to turbulent dissipation of the vortex cores. Their results revealed that the advection of these vortex pairs slowed down and lost their coherence since they ultimately underwent transition to turbulence. They also showed that the interaction between two adjacent jets altered their ultimate trajectories depending on the phase variation of these jets.
Gillaranz et al [10] studied the flow field of a synthetic jet actuator operating under mainly two different frequency and amplitude conditions. In their study, they utilized a high-frame-rate, digital PIV which they asserted to be the best measurement technique, because they managed to show the lack of instantaneous flow axisymmetry in the flow field. In addition to the experimental work, they presented a theoretical model for the oscillating flow field generated inside the actuator orifice.

They found that operation of actuator with a frequency of 10 Hz and membrane oscillation amplitude of 75 μm produces a net flow resembling a jet, while operation at 100 Hz with amplitude of 25 μm creates a net suction flow. They stated that the incidence of these attributes produced by the jet was not limited to these conditions and was observed to occur at any frequency between 10 Hz and 1000 Hz, depending on the magnitude of the oscillation of the membrane. They concluded that these two types of flow can be obtained with the same actuator simply by varying its operation conditions and this property of synthetic jets increases the potential applications of them to flow control.

Smith and Swift [26] studied experimentally two-dimensional synthetic jets of Reynolds numbers greater than 2000. They attempted to determine a threshold of the non-dimensional stroke length necessary for the jet formation of 2D synthetic jets using a slug-flow assumption. The calculation of the circulation from the centerline velocity indicated that the actual circulation was 15-30% greater than that obtained using the slug-flow assumption. Also, the formation threshold was not a constant and that the variations in the threshold are larger for the smaller nozzle sizes. Next, they compared the synthetic jets to continuous jets. They observed a resemblance to continuous jets in the far field.
However, in the near field the synthetic jet was observed to entrain more fluid, and thus grow faster than a continuous jet. They also investigated the effect of both the jet Reynolds number and non-dimensional stroke length. They concluded that the near field as well as the far field formation of the synthetic jet is a function of these dimensionless parameters.

Watson et al [43] studied experimentally the flow interactions between a pair of synthetic jets. They showed that there was a minimum spacing between actuators in order to produce a single coherent jet from each actuator in the array. Furthermore, they observed from the cross flow conditions that the combined effects of the yaw angle and the orifice spacing could either reduce or increase the amount of the coherent vorticity present in the flow. Therefore, they concluded that a desired phase differencing for flow control strategies could be achieved by altering these two parameters.

Finally, in their experimental study, Wu and Breuer [44] considered the application of the synthetic jet actuators to near wall turbulent boundary layer control. When the primary objective of one application required the actuator to generate high momentum flux, it is important that the synthetic jet actuator should operate at high Reynolds numbers and low Strouhal numbers. However, in their case, the actuator was operated in a very narrow range, characterized by low Reynolds number and high Strouhal number of the jet in order to influence the near wall boundary layer phenomena. They found that, in this particular range, the actuator changed from its reversible source-sink operation to a directed jet.

They also studied a pair of actuators placed closely. They observed that they interacted independently at low amplitudes, but interacted strongly at higher amplitudes.
1.3.2. Synthetic Jets to Control Separation Downstream of a Step

The ability to control a flow field, either actively or passively, in order to obtain a desired change is of immense technological importance (Gad-el-Hak [45]; Kral [46]). During the last decades, active flow control has been one of the leading technologies because of their ability to introduce energy into the flow.

Separated flows have always been the focus of experimental and computational research due to their practical importance (Gad-el-Hak and Bushnell [47]). The vast majority of the experimental studies involve separation from a sharp edge including backward-facing steps (Barkley et al [48]). Controlling of the separated flow downstream of a step is crucial for a mainly two reasons. First, separated flows produced by an abrupt change in geometry are of great importance in many engineering applications including MEMS. Second, the steady two-dimensional flow over a backward-facing step is an established benchmark in computational fluid dynamics.

Utilizing synthetic jets in order to reduce the separation bubble length or more importantly to eliminate the separation bubble has been considered by few researchers after the invention of synthetic jet actuators. The study of Rediniotis et al [31] conducted two flow separation control experiments in order to demonstrate the effectiveness of synthetic jet actuation in controlling the size of the separation region. The flow field generated in the backward facing step was studied through Particle Image Velocimetry (PIV). They also used low-order modeling of synthetic jet actuators via Proper Orthogonal Decomposition (POD) and Multiresolution analysis.

In their study, the synthetic jet actuator with an orifice width of 2 mm was placed on the step face to control the flow reattachment. The exact dimensions of their backward
facing step geometry are shown in Figure 1.4. They kept all the physical dimensions the same in both experiment and computation. They prescribed no-slip boundary conditions on the walls. For the computations, the inflow velocity profile was assumed as

\[ u_{\text{inflow}} = 10 \left( 1 - \frac{y}{h} \right)^2 \text{ cm / sec} \]  

(1.4)

whereas the jet velocity was assumed to be purely sinusoidal as

\[ u_{\text{Orifice}} = 40 \sin(2\pi f) \text{ cm / sec} \]  

(1.5)

They managed to reduce the size of the recirculation by an order of magnitude. The results obtained from experiments were in good agreement with computation. They noted the presence of small but dominant circulation right next to the orifice. They observed that the recirculation bubble existed always regardless of suction or blowing at the orifice.

Jain et al [32] studied numerically the flow over backward facing step with and without a synthetic jet using an incompressible Reynolds-Averaged Navier-Stokes
(RANS) solver. They performed a parametric study by varying jet velocity, frequency and its location in order to investigate the effect of these parameters in the recirculation region. In addition, they performed computations with two synthetic jets employed in the push mode on the step and the effect of their phase difference was studied.

They prescribed a parabolic velocity profile with maximum velocity of 10 cm/sec at the inlet of the domain and the synthetic jet was defined as a sinusoidal boundary condition at the orifice as in Rediniotis et al [31]:

\[ u_{\text{Orifice}} = 40 \sin(\omega t) \text{ cm/sec} \] (1.6)

No-slip boundary condition was prescribed on the walls.

They concluded that the presence of synthetic jet significantly reduced the reattachment length. They observed that more reduction in separation bubble size was achieved in full suction stroke than in full blowing stroke. They obtained good agreement with experiments of Rediniotis et al [31]. They found that increasing the frequency did not help reduce the reattachment length whereas higher velocity amplitude of the synthetic jet helped in reducing the separation region. Also, better results in terms of reducing the reattachment length was achieved when the synthetic jet was located on the step at a larger distance from the bottom wall.

In the presence of two synthetic jets, the best result was obtained as the lower jet (closer to the bottom wall) was led by an angle of 180° relative to the upper jet when compared to the cases of both 0° and 90°.
1.3.3. Synthetic Jets for Thermal Management

The study of improved heat transfer performance is referred to as heat transfer enhancement, augmentation, or intensification (Webb et al [49]). This has been an active research area of heat transfer technologies. It is a well known fact that the failure rate of electronic equipment strongly depends on their operating temperature. According to a survey in 1989 by the U.S. Air Force, more than 50% of all electronic failures are resulted from shortfall in effective thermal control (Fig. 1.5 (Yeh and Chu [50])). The purpose of thermal management is to achieve reliable temperature levels in electronic equipment in order to prevent thermal and mechanical failures.

![Pie chart showing major causes of electronic failures](image)

**Figure 1.5.** Major causes of electronic failures (Yeh and Chu [50]).

The demand for better and more effective cooling of electronic devices has been amplified largely for the last decades because of the microminiaturization in device sizes accompanied by higher power dissipation levels (Yeh and Chu [50]; ASM [51]; Bar-Cohen et al [52]). High-performance microdevices can dissipate power over 100 W.
This has been a primary and foremost concern for manufacturers as well as consumers of computers, cell phones, personal digital assistants and other electronic equipment. Figure 1.6 shows the 2004 update of the International Technology Roadmap for semiconductors (ITRS) (Prasher et al [53]). Furthermore, while a great deal of attention has been paid on thermal management of high-heat-flux systems, there is still a challenge for engineers and designers to attain more improved reliability objectives of thermal management for intermediate and low-heat-flux systems and equipment (Yeh and Chu [50]).

![Figure 1.6. ITRS roadmap(s) and CPU historical data for high-performance computers (Prasher et al [55]).](image)

Depending on the levels of device and system power dissipation, various cooling techniques have been developed in an attempt to maintain a desired thermal environment in which the microelectronic system can operate. Air cooling, either passive (via natural convection) or active (via forced convection), has been the most favored cooling technique because of its simplicity and serviceability. Passive air cooling is restricted to
low-heat-flux systems. In certain implementations, fans can be employed together with aluminum, magnesium or copper heat sinks in order to transport the dissipated heat to the environment. Although heat sinks are pervasively used for cooling of microdevices, their effectiveness is limited due to their high effective densities and their thermal conductivities being highly dependent on their material. One way to increase heat transfer of a heat sink is to increase the number of fins, but this reduces the spacing between them causing an increase in the flow resistance (Yeh and Chu [50]). Furthermore, it should also be noted that fan cooling may not be a desirable approach owing to the level of noise and system reliability. Liquid cooling is another option for cooling, particularly required in high-heat-flux systems due to its high heat convective rate and high heat capacity. Yet, despite its advantages, liquid cooling is more complex and stringent in terms of design, cost, manufacturing, operation and maintenance.

Utilizing synthetic jet actuators for thermal management of microelectronic devices is a relatively new technology. In their study, Campbell et al [36] investigated synthetic jets experimentally as a cooling device to remove the heat from a laptop processor. They first used synthetic jets to cool packaged thermal chips. They performed experiments with and without synthetic jet cooling. They also conducted experiments wherein the same package was cooled via natural convection using a pin fin heat sink to compare the effectiveness of synthetic jet cooling. They placed the synthetic jet above the package so that the created jet flow would impinge onto the center of the top surface of the package. They used two different orifice plates for the synthetic jet geometry. The square, single-hole orifice plate and the cut, multi-hole orifice plate designs are shown in Figure 1.7
where single-hole orifice has a diameter of 1.6 mm and multi-hole orifices have a diameter of 0.8 mm.

![Figure 1.7. Microjet orifice plate designs for laptop cooling; a) square, single-orifice plate b) cut, multi-orifice plate (Campbell et al [36]).]

They observed that there was an optimum distance between the package and the microjet. In addition, single-orifice microjet was found to be more effective than multi-orifice microjet. The synthetic jet cooling provided comparable results to heat sink cooling with natural convection. They achieved a reduction in processor operating temperature of 22% when compared to the laptop operating without synthetic jet cooling.

Mahalingam and Glezer [37] studied air-cooled plate fin heat sinks augmented with synthetic jet ejector arrays. They conducted experiments for three different configurations of cooling modules characterized with respect to power dissipation, thermal effectiveness and package volume and height. The configurations varied in positioning the synthetic jets relative to the heat sink in order to meet requirements of spatially constrained environments. Each fin of the heat sinks was straddled by a pair of synthetic jets that entrain cool ambient air upstream of the heat sink and eject it into the channels between the fins of the heat sink (Fig. 1.8) creating a secondary flow.
They showed that synthetic jets had potential to provide rapid, on-demand cooling. They achieved around 60-70% thermal effectiveness of synthetic jet heat sinks. They pointed out the importance of small-scale heat transfer and mixing induced by synthetic jets.

Kercher et al [54] studied miniaturized synthetic jets (microjets) as a device for thermal management of microelectronic devices. They investigated the cooling performance of the microjets in an open environment as well as in a vented and closed case environment. In their study, the microjet cooling device was placed above a thermal test die such that the synthetic jet removed heat via forced convection by impinging normal to the die surface. They represented an empirical characterization study through which they assessed the effect of geometrical parameters such as cooler to thermal test die spacing, membrane resonance frequency, orifice diameter and membrane driving
power. They calculated the relative temperature reduction by impinging synthetic jet as follows:

\[ R.T.R. = \frac{T_{\text{free}} - T_{\text{jet}}}{T_{\text{free}} - T_{\text{amb}}} \]  

(1.7)

Where \( T_{\text{free}} \) is the die temperature in case of natural convection, \( T_{\text{jet}} \) is the die temperature in case of forced convection via synthetic jet, and \( T_{\text{amb}} \) is the ambient temperature. They compared the results by forced convection with microjets to commercially available cooling fans and heat sinks and concluded that microjet cooling was compatible with cooling fans.

Beratlis and Smith [55] designed a synthetic jet actuator for the cooling of a one-dimensional VCSEL (Vertical Cavity Surface Emitting Laser) array (Fig. 1.9). They carried out a numerical study using a CFD code (FLUENT) to find the optimal performance of the synthetic jet for cooling. The flow was assumed to be compressible and fully turbulent. The moving membranes of the actuator cavity were modeled as planar velocity inlets with a spatially uniform velocity normal to the membrane and a sinusoidal oscillation in time with a velocity amplitude. No-slip boundary condition was prescribed on all other walls.

![Figure 1.9. A cross-sectional view of the prototype near the middle of one jet orifice](Beratlis and Smith [55])

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
They used an optimization procedure implementing MATLAB and FLUENT interactively in order to maximize the heat transfer from the VCSEL array. For an operating frequency of 350 Hz, three local optimum configurations were found with heat transfer rate of at least 10% larger than the initial prototype configuration.

Magalingham et al [56] implemented synthetic jets to study their effectiveness for electronic cooling at low flow rates. They conducted experiments and measured the flow using particle image velocimetry (PIV). They used one of the active heat sink configurations for high-power microprocessors that was developed by Mahalingam and Glezer [37]. The “synthetic jet ejector cooling module” is shown in Figure 1.10. The module consists of an aluminum plate-fin heat sink integrated with a synthetic jet actuator (plenum and driver) which straddles the L-shaped fins. Two arrays of synthetic jets (1f and 2f) are formed as a result of the time periodic pressure changes within the pendulum.

![Schematic diagram of a synthetic jet ejector heat sink module](image)

**Figure 1.10.** Schematic diagram of a synthetic jet ejector heat sink module a) side view b) top view (Mahalingam et al. [56]).
In their study, they discussed the effect of channel on the induced flow rate, power dissipated, heat transfer coefficient and thermal efficiency by using the simple configuration of two-dimensional synthetic jet ejectors in a rectangular channel. They found that the jet ejector resulted in higher heat transfer rates at low flow rates. The Nusselt numbers for synthetic jet ejector flows were calculated to be six to eight times higher than for comparable conventional turbulent flow.

Another study to evaluate the effectiveness of the synthetic jet was performed by Timchenko et al [35]. They studied numerically the possibility of utilizing synthetic jets to enhance heat transfer rates in microchannels. They considered a two-dimensional channel with a height of 200 \( \mu \text{m} \) and length of 4.2 mm. They prescribed isothermal boundary condition on the top wall and adiabatic one on the bottom wall of the microchannel. The Knudsen number was calculated to be less than 0.01; therefore the continuum approach was still valid. They simulated the flow field using a commercially available CFD package (CFX-5.6) which solved unsteady compressible laminar flow. The displacement of the membrane was defined as sinusoidal function:

\[
Y_m = A\left(1 - \left(\frac{x}{r}\right)^2\right)\sin(2\pi ft)
\]  

(1.8)

They evaluated the effectiveness of the cooling by comparing the heat transfer rates with and without synthetic jets. They managed to obtain improved cooling and observed that the effectiveness was highly dependent on the placement of the cooling device.

1.3.4. Other Applications of Synthetic Jets

Smith and Glezer [20] studied, experimentally, the flow characteristics of high aspect ratio synthetic jets and their application to thrust vectoring and direct excitation of small
motions in a conventional rectangular air jet. They observed from Schlieren visualization of a single synthetic jet that successive vortex pairs that were formed during each cycle of the membrane motion did not undergo pairing but rather broke down and lost their coherence structure at distances greater than $10\delta$. Then they utilized millimeter-scale, high aspect ratio jets placed near a primary conventional jet one to two orders of magnitude larger. Also, the operating frequency of the actuators was 1000 Hz which was an order of magnitude higher than the natural unstable frequency of the primary jet. They observed that the primary jet was vectored either towards or away from the smaller jets at angles ranging between 30°-80°. They, therefore, concluded that the actuator motion increased the small scale motions within the primary jet, thus suggesting enhanced mixing in the shear layer.

Przekwas et al [57] studied the microfabricated arrays of synthetic jets. Their study represents two complementary techniques: High-fidelity CFD-based model and a reduced model. They demonstrated that a reduced dimensionality of a synthetic jet using a single control volume or a 1D model can be used to model large number of jets in an accurate and cost effective manner.

Mautner 1999 [33] investigated the application of synthetic jets to the low Reynolds number, two-dimensional channel flows in biosensor microfluidic systems. For their numerical simulations, they utilized a hybrid approach of Lattice Boltzman method for flow field computations and a finite difference convection-diffusion equation for passive scalar transport. In their study, although they modeled the synthetic jet with a cavity, they replaced the synthetic jet’s time-dependent membrane oscillation with so-called wall jets which was characterized by a time-dependent wall velocity as follows:
29

\[ V_{\text{wall}}(x,t) = A_{\text{wall}}(t) \sin(\omega t) \]  \hspace{1cm} (1.9)

They implemented wall jets with various cavity-orifice geometries as well as varying wall velocities. The results for channel flow with single and double wall jets indicated that the cavity-orifice geometry did not have significant effect on the resulting flow field. In addition, the simulations showed that the implementation of wall jets imparted momentum to the channel flow and created significant areas of fluid rotation. They stated a similarity of the wall jets in concept to synthetic jets; however, the wall jets provided a non-zero mass flow unlike synthetic jets.

McCormick [30] used a new concept of synthetic jet actuator, the so-called “directed synthetic jet”, with a throat curved in the downstream tangential direction. They investigated numerically the directed synthetic jets for boundary layer separation. On the in-stroke of the throat velocity, vertical momentum is imparted to the flow causing the neck to ingest the approaching low momentum fluid. On the out-stroke of the throat velocity, the fluid particles are re-accelerated and injected with positive axial momentum into the wall region of the boundary layer. By doing so, the ability of the boundary layer to resist separation was increased.

Mittal et al [8] studied the interaction of a two dimensional synthetic jet interacting with a flat plate boundary layer using an incompressible Navier-Stokes solver. They modeled the membrane as a moving boundary in an effort to compute the internal cavity flow accurately. A parametric study was carried out where the membrane amplitude, the boundary layer Reynolds number based on the boundary layer thickness and the orifice width to height ratio were varied while all the other parameters were kept constant.
The simulations showed that the presence of an external flow had significantly different flow in terms of vortex dynamics and the jet velocity profiles when compared to the quiescent external flow case. They also characterized the jet in terms of the moments of the jet profile. They examined the ingestion and the expulsion strokes separately since they revealed a distinctly different jet behavior. Finally, they investigated the aero-shaping effect of the synthetic jet. They found that large mean recirculation bubbles were formed in the external boundary layer only if the jet velocity is significantly higher than the external flow velocity.

Mittal and Rampunggoon [58] studied the ability of synthetic jets to form large mean recirculation zones in the presence of an external flow. For the computations they used an incompressible Navier-Stokes solver neglecting the compressibility effects within the actuator cavity. They suggested the so-called virtual aeroshaping effect of the synthetic jet to be one mechanism to affect separation reduction. They observed that the thickness of the boundary layer relative to the slot width ($\delta/d$) and the jet velocity relative to the external flow velocity ($\bar{V}_j/U_\infty$) had significant impact on the bubble size. They also explored the scaling of the bubble size with the momentum coefficient defined as $C_\mu = \bar{V}_j^2d/U_\infty^2\theta$ and concluded that the normalized bubble length ($L_\theta/d$) grew linearly with the momentum coefficient $C\mu$. Also, the thickness of the boundary layer relative to the slot width found to have a significant impact on the bubble size.

Lockerby and Carpenter [13] modeled an alternative mode of the synthetic jet actuator for use in active turbulence control. The membrane was modeled using classical thin-plate theory, with the stiffness of the attached piezo-device incorporated and clamped at its edges. The fluid motion in the cavity was not modeled for numerical
economy. At the center point of the membrane, symmetry condition was enforced. $Kn$ number based on the orifice diameter was calculated as 0.0065, thus continuum criterion was satisfied allowing the Navier-Stokes equations to be used.
CHAPTER II

PHYSICS of SLIP FLOW REGIME

In this chapter, we present the basic equations and their boundary conditions for gas flows in slip flow regime after a general discussion of micro flows.

2.1. MEMS and Microfluidics

One of the most promising and exciting technological developments of the last decade of the 20th century was the field of micro-electro-mechanical systems (MEMS). The development of miniaturized non-electronic devices lagged behind the miniaturization of microelectronics where electrical devices like transistors, diodes and resistors are integrated to give an electronic system (Nguyen and Werely [5]; Koch et al [59]). The milestone events in microelectronics were the invention of the transistor in 1947 and the development of the integrated circuit (IC) in 1958. The micromachining technology emerged in the late 1980s providing micron-level transducers (Ho and Tai [15]); then, these devices were integrated with electronic, fluidic and optical components to form MEMS. MEMS technology allows low cost systems with wide variety of applications spanning from aerospace control systems, to advanced energy systems, to biomedical and biochemical devices owing to its capability to offer many standard functions with reduced size, mass and power requirements (Beskok [17], Janson et al [60]).

MEMS refer to devices that have a characteristic length of less than 1 mm but more than 1 μm, that combine electrical, mechanical, optical and fluidic components.
and that are fabricated using integrated circuit batch-processing technologies (Gad-el-Hak [61]). However, it is suggested in (Nguyen and Werely [5]) that it would be more appropriate to use micro system technology (MST) as the term for the microtechnology that emerges today with its fluidic and optical components.

A well-formed understanding of MEMS requires an established knowledge of the materials used to build the devices, due to the fact that the material properties of each element which provides a critical function in a device can influence system performance. MEMS technology covers all range of microelectronic fabrication methods, allowing a final integration of micromechanics and microelectronics. Novel techniques have been added where materials different than silicon need to be used. The list materials used in fabrication of MEMS devices are too long and it is out of the scope of this study. Therefore, readers are referred to comprehensive reviews presented in (Sharpe [62]; Zorman and Mehregany [63]). Also, detailed information on fabrication techniques of MEMS can be found in Nguyen and Werely [5], Koch et al [59], Madou [64], and Ho and Tai [65].

Not all MEMS devices do involve fluid flows. Microducts, micronozzles, micropumps, microturbines, microvalves, and microflow sensors are examples of MEMS devices involving fluid flow. The term “microfluidics” refers to a new discipline that pertains to the design of systems in which small volumes of fluids brings about changes in fluid behavior. Microfluidic systems can be used for various applications that span drug dispensing, ink-jet printing and general transport of liquids, gases and their mixtures (web site [66]). For MEMS, the device size should be submillimeter scale whereas for microfluidics only the space where the fluid is
processed has to be miniaturized regardless of the device size. Hence, a microfluidic device can be identified by the fact that it has one or more channels with at least one dimension less than 1 mm (web site [67]) and can also be utilized for targeting some structures in macron-level flows.

Micro devices are subject to behave differently than the objects we are used to handling in our daily life. Therefore, the operation of microfluidic devices cannot always be predicted from conventional flow models owing to the unfamiliar physics of fluids involved in micro flows (Gad-el-Hak [3]). The inertial forces, for example, tend to be quite small and surface effects tend to dominate the behavior of these small systems. Friction, electrostatic forces, and viscous effects due to the surrounding air or liquid become considerable as the devices become smaller. Surface effects dominate micro flows. The surface-to-volume ratio for a machine with a characteristic length of 1 m is $1 \text{ m}^{-1}$, while that for a MEMS device having a size of 1 μm is $10^6 \text{ m}^{-1}$. This increase substantially affects the transport of mass, momentum, and energy through the surface. The small length scale of microdevices may invalidate the continuum approximation altogether. Slip flow, thermal creep, rarefaction, viscous dissipation, compressibility, intermolecular forces and other unconventional effects may have to be taken in to account, preferably using only first principles such as conservation of mass, Newton’s second law, and conservation of energy.

Under macro scale continuum conditions it is possible to have creeping flows with a small Reynolds number, Re and a small Mach number, $Ma$. However, in the case of micro flows the Re number is small due to small length scales of the micro
device rather than very small velocities. As a result, higher $M$ numbers can be achieved in micro flows compared to the creeping continuum flows. The main differences between fluid mechanics at macro scales and in micro domain can be broadly classified into four areas (Karniadakis and Beskok [2]): Non-continuum effects, Surface-dominated effects, Low Reynolds number effects, and Multi-scale, multi-physics effects.

When considering these differences one is faced with the question of which model to use and which boundary condition to impose in order to obtain a solution to a problem in micro domain. Moreover, the answers to these questions strongly depend on whether the fluid is a gas or a liquid. Hence, the behavior of gas flows and liquid flows in micro devices will be discussed separately in the following parts.

2.2. Flow Modeling

2.2.1. Intermolecular Forces

All three states of matter, solids, liquids, gases, are comprised of molecules that are all interacting with each other (Nguyen and Werely [5]; Gad-el-Hak [3]). This interaction as well as the behavior of all states depends on the forces between the molecules. Neutral atoms and molecules are subject to two distinct forces in the limit of large distance, and short distance: an attractive van der Waals (or dispersion) force and a repulsion force, the result of overlapping electron orbitals, referred to as Pauli repulsion (web site [68]). Van der Waals forces originally referred to all intermolecular forces, but it is now more commonly used to refer to those forces.
which arise from the polarization of molecules into dipoles. This includes forces that arise from fixed or angle-averaged dipoles (Keesom forces) and free or rotation dipoles (Debye forces) as well as shifts in electron cloud distribution (London forces).

The interaction of two non-reacting, non-ionized molecules is represented by Lennard-Jones potential which incorporates the shape effects by an anisotropic repulsive core and anisotropic dispersion interactions as follows:

\[
V_{ij}(r) = 4\varepsilon \left[ c_{ij} \left( \frac{r}{\sigma} \right)^{-12} - d_{ij} \left( \frac{r}{\sigma} \right)^{-6} \right] 
\]

(2.1)

Where \( \varepsilon \) is a characteristic energy scale and \( \sigma \) is a characteristic length scale, \( r \) is the distance separating the molecules \( i \) and \( j \), \( c_{ij} \) and \( d_{ij} \) are parameters particular to the pair of interacting molecules. These parameters can be fitted to reproduce experimental data or deduced from results of accurate quantum chemistry calculations. The \( r^{-12} \) term describes the repulsive force that exists between two molecules when they are brought together and the \( r^{-6} \) term describes the mildly attractive potential due to van der Waals force between any two molecules.

The force between two interacting molecules can be obtained simply by the derivative of the potential energy between those molecules, \( V_{ij} \), as:

\[
F_{ij} (r) = -\frac{\partial V_{ij}(r)}{\partial r} = \frac{48\varepsilon}{\sigma} \left[ c_{ij} \left( \frac{r}{\sigma} \right)^{-13} - \frac{d_{ij}}{2} \left( \frac{r}{\sigma} \right)^{-7} \right] 
\]

(2.2)

For an appropriate choice of the parameters in the equation, a reasonable description of real fluids is possible. For example, using \( \varepsilon/k_B = 120 \text{K} \), where \( k_B \) is the Boltzmann's constant, \( \sigma \approx 0.34 \text{nm} \), and \( c_{ij} = d_{ij} = 1 \), Lennard-Jones potential
provides reasonably accurate results for liquid argon. The characteristic time scale of molecular interactions for Lennard-Jones potential is given by

\[ \tau = \sqrt{\frac{m\sigma^2}{\varepsilon}} \]  

which is the period of oscillation about the minimum in the Lennard-Jones potential. The \( m \) represents the mass of a single molecule.

The atoms or molecules that comprise the solid are packed close together and these constitutive elements have fixed positions, with a mean intermolecular spacing of about \( \sigma \), in space relative to each other. The solid molecules interact with each other through Lennard-Jones force. This accounts for the solid’s rigidity. As the solid matter is heated, the average molecular thermal energy becomes high enough so that the solid enters the liquid phase which enables molecules to pass each other freely from one to another. The molecules of a liquid are still relatively close to each other, approximately \( \sigma \). Further heating of the matter will break the forces between the molecules keeping them together or pushing them apart. This results in the gas phase which is a completely disordered state of the matter. The mean space between the molecules of a gas matter is as large as \( 10\sigma \). The only interaction between gas molecules are rare and random collisions. The Lennard-Jones forces become significant when compared with their kinetic energy, only when the molecules are close to each other during brief collisions.
2.2.2. Continuum Approach

A flow field can be modeled either as the fluid really is or as a continuum. Continuum mechanics is a branch of physics (specifically mechanics) that deals with continuous matter, including both solids and fluids. The study of fluid mechanics at conventional scales generally proceeds from the assumption that the fluid can be treated as a continuum.

A fluid matter is composed of molecules and it commonly has some sort of heterogeneous microstructure. This molecular nature of the fluid is left out of count in continuum assumption and considered to be continuous medium. In this sense, the physical quantities, such as density, velocity, pressure and temperature, can be handled in infinitesimal limit and are assumed to be defined at every point as a function of time and space changing continuously from one point to another (Nguyen and Werely [5]; Gad-el-Hak [3]). Differential equations can thus be employed in solving continuum mechanics problems. Some of these differential equations are specific to the materials being investigated and are called constitutive equations, while others capture fundamental physical laws, such as conservation of mass or conservation of momentum. Additionally, the flow must not be too far from thermodynamic equilibrium. When this is the case, the validity of the continuum equations becomes questionable and one must use the statistical or molecular approaches to model the flow domain more accurately.

The transport quantities such as viscosity and diffusivity must be continuous when treating the fluid as continuum. For the transport quantities to behave continuously, it is important that the fluid molecules interact much more often with
themselves than with flow boundaries. Hence, the time and length scales associated with the intermolecular collisions are important parameters for many applications.

For gases, the best estimate of an interaction length scale is known as the mean free path, \( \lambda \), which is defined as the average distance a particle travels between two successive collisions with other particles. For a liquid, the molecules are fundamentally in a continuous interaction, so the mean free path is not a good estimate of the interactions occurring in a space. For a simple gas of hard spherical molecules in thermodynamic equilibrium the mean free path is given in the following form:

\[
\lambda = \left( \frac{2}{\pi} \frac{d^2 n}{\rho} \right)^{-1}
\]  

(2.4)

Where \( n \) is the number density of molecules and \( d \) is the molecular diameter. The gas molecules are traveling with speeds proportional to the speed of sound. The "mean-square molecular speed" of gas molecules is given as:

\[
\bar{c} = \sqrt{\frac{3p}{\rho}} = \sqrt{3RT}
\]  

(2.5)

Here, \( \rho \) is the density, \( p \) is the pressure, \( T \) is the temperature and \( R \) is the specific gas constant in the form:

\[
R = \frac{\bar{R}}{M} \quad \text{where} \quad \bar{R} = 8.3145 \text{ kJ/kmol·K}
\]  

(2.6)

In the above equation \( \bar{R} \) is the universal gas constant and \( M \) is the molar mass of the gas.

The governing equations of continuum mechanics are presented by Navier-Stokes equations. They should be used, therefore, as long as the flow is not too far from thermodynamic equilibrium.
2.2.3. Kinetic Gas Theory

Kinetic theory attempts to explain macroscopic properties of gases, such as pressure, temperature, or volume, by considering their molecular composition and motion.

Gases consist of mostly space with a few molecules colliding only infrequently. Consequently, they can be described quite well by the kinetic gas theory in which a gas molecule is considered to move in a straight line at a constant speed until it strikes another molecule—which it does infrequently. Many of the conclusions of kinetic gas theory are presented in this section. The equation of state for a dilute gas is the ideal gas law which can have several equivalent forms. One of these forms is:

\[ p = n k_B T \]  \hspace{1cm} (2.7)

Where \( k_B \) is Boltzmann’s constant \((k_B = 1.3805 \times 10^{-23} \text{ J/K})\). Therefore, for most of the gas micro flow applications, the number density of the molecules at a certain pressure and temperature can be calculated using Eq. (2.7).

Under standard conditions, the number density of any gas is \( n = 2.7 \times 10^{25} \text{ m}^{-3} \). Assuming that all the molecules of the gas is packed uniformly, then the mean molecular spacing can be obtained as

\[ \delta \equiv n^{-\frac{1}{3}} = 3.3 \times 10^{-9} \text{ m} \]  \hspace{1cm} (2.8)

Comparing the mean molecular spacing and a molecular diameter of a typical gas, for example \( N_2 \), we obtain an order of magnitude difference:

\[ \frac{\delta}{d} \approx 10 > 1 \]  \hspace{1cm} (2.9)
Gases that hold this equation are called \textit{dilute gases}. For dilute gases, binary collisions are assumed for the intermolecular interaction and simultaneous multiple molecule collisions are not likely.

Several transport quantities are considered in gas dynamics. The mean free path, \( \lambda \), and the mean-square molecular speed, \( \bar{c} \), were given by Eq. (2.4) and (2.5), respectively, in the previous section. Another important quantity to determine the type of the flow that will develop in the gas is the \textit{speed of sound} and it is given by,

\[
a = \sqrt{\gamma RT}
\]

where \( \gamma \) is gas constant which is defined as the ratio of specific heat at a constant pressure, \( C_p \) to the specific heat at a constant volume \( C_v \). One more quantity of importance is the \textit{viscosity} of the gas. The viscosity is given according to the kinetic gas theory by the following relationship:

\[
\nu = \frac{\mu}{\rho} = \frac{1}{2} \frac{\lambda \bar{c}}{P}
\]

In addition to the transport quantities, two important dimensionless parameters in fluid mechanics are the \textit{Reynolds number}, \( Re \) and the \textit{Mach number}, \( Ma \). The Reynolds number is the ratio of inertial forces to viscous ones:

\[
Re = \frac{UL}{\nu}
\]

Where \( U \) is a \textit{characteristic velocity}, \( L \) is a \textit{characteristic length}, and \( \nu \) is the \textit{kinematic viscosity} of the fluid. Flow patterns are typically functions of the Reynolds number. The different flow behavior can be observed depending on the order of the Reynolds number. For example, when \( Re \approx 1 \), viscous effects in the flow will be comparable to the inertial effects; for values of \( Re<<1 \), viscous effects will dominate.
the flow behavior whereas the inertial forces will dominate the viscous effects for
Re$\gg$1. The Mach number is the ratio of inertial forces to elastic ones and can be
considered as a measure of the compressibility of the gases:

$$Ma = \frac{U}{a} \quad (2.13)$$

Flows can be classified as subsonic, sonic and supersonic for $Ma<1$, $Ma=1$ and
$Ma>1$, respectively. Similarly, for values of $Ma>0.3$ the flow must be treated as
compressible. Although necessary, this assumption is not a sufficient for a flow is
experiencing large pressure changes. Corresponding to the pressure changes would
be substantial density changes that must be taken into account when writing the
equations of motion of the flow. Under certain conditions, the density of the gas flow
does not change substantially and the flow is then called incompressible. Note that a
fluid flow can be considered incompressible even though the fluid itself is still
compressible.

2.2.4. Flow Regimes in Microfluidics

From the continuum point of view, liquids and gases are both fluids complying
with the same equations of motion. Modeling of liquids and gases in micro domain,
however, require different approaches. In the present study, the working fluid is
chosen to be a diatomic gas (Nitrogen, $N_2$), therefore the focus will be essentially on
the flow phenomena in gas micro flows and their modeling. Yet, a brief explanation
about liquid micro flows will also be mentioned in the following subsections.
2.2.4.1. Gas Micro Flows

A key non-dimensional parameter for gas micro flows is the Knudsen number, which is defined as

\[ Kn = \frac{\lambda}{L} \]  \hspace{1cm} (2.14)

where \( \lambda \) is the mean free path and \( L \) is the characteristic length of the geometry. Mean free path is known as the average distance a particle travels between two successive collisions with other particles. The characteristic length \( L \) can be some overall dimension of the flow, but more precise choice is the scale of the gradient of a macroscopic quantity, as, for example, the density \( \rho \) (Gad-el-Hak [3]),

\[ L = \frac{\rho}{\frac{\partial \rho}{\partial y}} \]  \hspace{1cm} (2.15)

The Knudsen number is related to the Re and \( M \) numbers as follows:

\[ Kn = \sqrt{\frac{\gamma \pi M}{2 \ Re}} \]  \hspace{1cm} (2.16)

In small-scale world, non-continuum, rarefied gas flow phenomena become prevailing, which may make MEMS behave rather differently from their counterparts at macro scales (Fan and Shen [69]). Rarefied gas flows are also encountered in flows in low-pressure applications such as high-altitude flying and high-vacuum gadgets. The local value of \( Kn \) number in a particular flow determines the degree of rarefaction and the degree of validity of the continuum model (Gad-el-Hak [3]; Beskok [17]). In the limit of zero Knudsen number, the transport terms in the continuum momentum and energy equations are negligible and the Navier-Stokes equations then reduce to the inviscid Euler equations. As \( Kn \) increases, rarefaction
effects become more important, and thus pressure drop, shear stress, heat flux and corresponding mass flow rate cannot be predicted from flow and heat transfer models based on continuum hypothesis. A classification of the different flow regimes depending on the range of the Knudsen number is given in (Gad-el-Hak [3]) as follows:

- Continuum flow regime for $Kn \leq 10^3$
- Slip flow regime for $10^3 < Kn < 0.1$,
- Transition flow regime for $0.1 < Kn < 10$.
- Free-molecular flow regime for $Kn \geq O(10)$.

It should be noted that the ranges for continuum and slip flow regimes differs from those in (Karniadakis and Beskok [2]) where the continuum flow regime is defined for $Kn < 10^{-2}$ and the slip flow regime is defined for values of $Kn$ number range of $10^{-2} \leq Kn < 0.1$. Also, the classification of the flow regimes according to the Knudsen number is based on empirical information and thus limits between the different flow regimes may depend on the flow geometry. The operation regimes for typical MEMS devices at standard conditions, based on the smallest characteristic flow dimensions, are shown in Figure 2.1.

In slip flow regime, we use Navier-Stokes equations with modified boundary conditions at the solid surface. The fundamental assumption in this analysis is that the dynamic similarity exists between rarefied flows in a low-pressure environment at macro scales and gas flows at micro scales (Karniadakis and Beskok [2]; Liou and Fang [4]). This assumption is justified theoretically based on the analysis of the Boltzmann equation for small Re number and $Kn$ number. Based on this assumption
gas micro flows are simulated subject to slip boundary conditions. In order to demonstrate the effects of rarefaction and slip in micro geometries we compare and contrast these findings with the continuum no-slip solutions whenever possible. Higher-$Kn$ analyses require higher-order representations or a Direct Simulation Monte Carlo (DSMC) (Mavripilis et al [71]; Oran et al [72]). The DSMC method is suitable for flows with very high speeds and low pressures. With this method, feasible simulations at room temperature and pressure conditions with low velocities require improved or novel analysis techniques. So-called the IP-DSMC technique (Fan and Shen [69]) has been shown to reduce computer requirements considerably compared to DSMC. The Lattice-Boltzmann method is another very attractive candidate for micro-fluidic simulations (Chen and Doolen [73]). However, when applicable, Navier-Stokes solvers are among the most efficient. Classification of fluid modeling is depicted in Figure 2.2.

![Figure 2.1. Characteristic lengths of typical fluidic microsystems, with the range of Knudsen number corresponding to standard conditions (Karniadakis and Beskok [70]).](image-url)
In gas micro flows, we encounter four important effects, namely

- Rarefaction,
- Compressibility,
- Viscous heating,
- Thermal creep.

In the slip flow regime, the competing effects of compressibility and rarefaction are observed. Fluid acceleration in a long microchannel where the entrance pressure is atmospheric and the exit conditions are near vacuum affects the pressure gradient along the channel resulting in a nonlinear pressure distribution. Curvature in pressure distribution is due to compressibility effects and it increases with increased inlet to outlet pressure ratio across the channel. The effect of the rarefaction is to reduce the curvature in the pressure distribution. The viscous heating effects are due to the work done by viscous stresses (dissipation), and they are important for micro flows, especially in creating temperature gradients within the domain even for isothermal
surfaces. The thermal creep (transpiration) phenomenon is a rarefaction effect. For a rarefied gas flow it is possible to start the flow with tangential temperature gradients along the channel surface. In such a case the gas molecules start creeping from cold toward hot.

The flow of a fluid through a microfluidic channel can be characterized by the Reynolds number. For many microchannels, $L$ is equal to $4A/P$ where $A$ is the cross sectional area of the channel and $P$ is the wetted perimeter of the channel. Due to the small dimensions of microchannels, the Re is usually much less than 100, often less than 1.0. In this Reynolds number regime, flow is completely laminar and no turbulence occurs. The transition to turbulent flow generally takes place in the range of Reynolds number 2000. Laminar flow provides a means by which molecules can be transported in a relatively predictable manner through microchannels. Note, however, that even at Reynolds numbers below 100, it is possible to have momentum-based phenomena such as flow separation (web site [74]).

2.2.4.2. Liquid Micro Flows

The behavior of liquids is completely different from that of gases and remarkably more complex. Liquid molecules act in a state of constant collision and it is difficult to predict the molecular effects in liquids. As a result, the molecular theory of liquids has not been as well developed as the kinetic theory for gases.

Liquids are essentially incompressible and have viscosity and density that are some orders of magnitude higher than those of gases. The density of liquids is about 1000 times the density of gases, the distance between molecules in liquids is
approximately one order of magnitude less than that in gases and they certainly do not have a mean free path. Thus, there are no parameters to help in establishing the behavioral regime of the liquids as there is Knudsen number determining the gas flow regimes.

Several constitutive relationships are needed to close the conservation equations governing the liquid flow. Two primary assumptions are that the liquid is Newtonian and that the heat transfer is proportional to the temperature gradient. For liquids comprised of a single type of simple molecules these assumptions are quite valid. Liquids are treated as continuum whenever the characteristic length scale is larger than 10 nm. Therefore in most micro flows, a continuum description suffices where Navier-Stokes equations can be used to solve the problems. However, in submicron dimensions the flow may behave discontinuously and atomistic modeling such as Molecular Dynamics (MD) is required (Karniadakis and Beskok [2]; Gad-el-Hak [3]; Kamiadakis et al [70]; Sharp et al [75]).

The phenomena encountered in fluid flows are listed in (Karniadakis and Beskok [2]) as:

- Non-Newtonian behavior in the near-wall region,
- Wetting,
- Adsorption,
- Electrokinetics.

In the near wall region, the fluid atoms are organized in horizontal layers parallel to the atomic layers. This layering phenomenon is responsible for large density fluctuations very near to the wall. While in liquids this effect extends only a few
atom diameters from the wall, in gases the wall-fluid interaction extends over much greater length and this has to be accounted for explicitly. At high shear rates and even for Newtonian fluids the liquid behavior in the near wall vicinity is found to be non-Newtonian.

The _wetting_ of the solid surfaces by liquids can be exploited in microfluidics to determine precisely defined routes based on surface tension gradients and different types of surfaces, such as hydrophobic or hydrophilic. Wetting may also affect flow and performance of a MEMS device by altering its mechanical response or even blocking flow channels.

_Adsorption_ is important in interactions of fluids with nanoporous materials such as glasses. The boundary conditions for fluid flow are very sensitive to the type and amount of adsorption on the walls of a MEMS device. _Electrokinetic effects_ are also important in microfluidics and provide means of maintaining a certain level of flow rate with practically uniform profiles.

**2.3. Slip Flow Regime**

In the continuum flow regime, the Navier-Stokes equations with no-slip boundary conditions govern the flow. Yet, for flows with Knudsen numbers greater than 0.1, a sublayer of the order of one mean free path, known as the _Knudsen layer_, starts to become dominant between the bulk of the fluid and the wall surface. The flow in the Knudsen layer cannot be analyzed with Navier-Stokes equations and necessitates special solutions of the Boltzmann equation. However, for \( Kn \leq 0.1 \), the Knudsen layer accounts for less than 10% of the channel height (or the boundary thickness for
external flows), and this layer can be neglected by extrapolating the bulk gas flow towards the walls. This results in a finite velocity slip at the wall and the corresponding flow regime is known as the slip flow regime. In the slip flow regime, the flow is governed by the Navier-Stokes equations, and rarefaction effects are modeled through the partial slip at the wall using Maxwell's velocity slip and von Smoluchowski's temperature jump boundary conditions (Gad-el-Hak [3]; Beskok [17; Agarwal and Yun [76]).

2.3.1. Governing Equations

The three primary conservation laws that are used to model the thermodynamic flow problems are the conservation of mass, momentum and energy by applying the principles of mechanics and thermodynamics. These approaches have been discussed greatly in fluid mechanics textbooks (Fox and McDonald [77]; White [78]), hence will only be summarized here.

Consider a fluid flow in the non-deformable control volume $\Omega$ bounded by the control surface $\partial \Omega$ with $\vec{n}$ the unit outward normal. The equations of motion can be derived in an absolute reference frame and can be formulated either in integral or differential forms. These governing equations in integral form are,

\[
\frac{\partial}{\partial t} \int_{\Omega} \rho d\Omega + \int_{\Omega} \rho \vec{v} \cdot \vec{n} dS = 0 \tag{2.17}
\]

\[
\frac{\partial}{\partial t} \int_{\Omega} \rho \vec{v} d\Omega + \int_{\Omega} \rho (\vec{v} \cdot \vec{n}) - \vec{n} \sigma \cdot dS = \int_{\Omega} \vec{f} d\Omega \tag{2.18}
\]

\[
\frac{\partial}{\partial t} \int_{\Omega} E d\Omega + \int_{\Omega} (E \vec{v} - \sigma \vec{v}) \vec{n} dS = \int_{\Omega} \vec{f} \cdot \vec{v} d\Omega \tag{2.19}
\]
where \( \vec{v}(\vec{x},t) = (u,v,w) \) is the velocity field in the Cartesian coordinate system \((x, y)\), \(\rho\) is the density, \(\vec{q}\) is the heat flux vector, and \(\vec{f}\) represents the external forces acting on the control volume. Also, \(E = \rho(e + 1/2\vec{v} \cdot \vec{v})\) is the total specific energy where \(e\) represents the internal specific energy. These equations can be cast in differential form by using Gauss’ theorem:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \tag{2.20}
\]

\[
\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot [\rho \vec{v} \vec{v} - \sigma] = \vec{f} \tag{2.21}
\]

\[
\frac{\partial E}{\partial t} + \nabla \cdot [E \vec{v} - \sigma \vec{v} + \vec{q}] = \vec{f} \cdot \vec{v} \tag{2.22}
\]

These equations comprises of five partial differential equations (one for mass, three for momentum, and one for energy) in 17 unknowns. Thus, they do not present a well-determined system of equations. To close these equations, a necessary but not sufficient condition for their solution, we must look to constitutive relationships and constitutive thermodynamic properties of the fluid. For Newtonian fluids, the stress tensor consists of the normal components and the viscous stress tensor and is a linear function of the velocity gradient,

\[
\sigma = -p\vec{I} + \tau \tag{2.23}
\]

where,

\[
\tau = \mu[\nabla \vec{v} + (\nabla \vec{v})^T] + \zeta (\nabla \cdot \vec{v}) \vec{I} \tag{2.24}
\]

Here \(\vec{I}\) is the unit tensor, and \(\mu\) and \(\zeta\) are the first and second coefficients of viscosity. They are related by the Stokes hypothesis which expresses local thermodynamic equilibrium:
\[ 2\mu + 3\zeta = 0 \quad (2.25) \]

The heat flux vector is related to temperature gradients via the Fourier diffusion law as follows:

\[ \bar{q} = -k \nabla T \quad (2.26) \]

In the above equation, \( k \) is the thermal conductivity which, in some cases, is a function of temperature.

Finally, an equation of state is required to form a well-determined system. For ideal gases, it was given by Eq. (2.7). An equivalent form of this equation can be written as follows:

\[ p = \rho RT \quad (2.27) \]

The system of equations (2.20), (2.21), (2.22) and (2.27) is called compressible Navier-Stokes equations. These equations can be rewritten in two-dimensional form as follows:

\[
\begin{aligned}
\frac{\partial}{\partial t} \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ E \end{pmatrix} &+ \frac{\partial}{\partial x} \begin{pmatrix} \rho u \\ \rho u^2 + p + \sigma_{xx} \\ \rho uv + \sigma_{xy} \\ (E + p + \sigma_{xx}) \cdot u + \sigma_{xy} \cdot v + q_x \end{pmatrix} \\
&+ \frac{\partial}{\partial y} \begin{pmatrix} \rho v \\ \rho uv + \sigma_{yx} \\ \rho v^2 + p + \sigma_{yy} \\ (E + p + \sigma_{yy}) \cdot v + \sigma_{yx} \cdot u + q_y \end{pmatrix} = 0
\end{aligned}
\]
2.3.2. Boundary Conditions

In the slip flow regime, the compressible Navier-Stokes equations are solved subject to the velocity slip and temperature boundary conditions given by

\[ u_s - u_w = \frac{2 - \sigma_v}{\sigma_v} \frac{1}{\rho(2RT_w / \pi)^{1/2} \tau_s} + \frac{3}{4 \gamma \rho R T_w} \frac{\operatorname{Pr}(\gamma - 1)}{\gamma \rho R T_w} (-q_n) \]  \hspace{1cm} (2.29)

\[ T_s - T_w = \frac{2 - \sigma_T}{\sigma_T} \left[ \frac{2(\gamma - 1)}{\gamma + 1} \right] \frac{1}{R \rho(2RT_w / \pi)^{1/2} (-q_n)} \]  \hspace{1cm} (2.30)

where \( \gamma \) is the ratio of specific heats, \( \rho \) is the density and \( R \) is the specific gas constant; \( q_n \) and \( q_s \) are the normal and tangential heat-flux components and \( \tau_s \) is the viscous stress component pertaining to the skin friction; \( u_w \) and \( T_w \) are the reference wall velocity and temperature, respectively; \( \operatorname{Pr} \) is the Prandtl number and it is given by,

\[ \operatorname{Pr} = \frac{C_p \mu}{k} \]  \hspace{1cm} (2.31)

Also, \( \sigma_v \) and \( \sigma_T \) are the tangential momentum and energy (thermal) accommodation coefficients that determine the effectiveness of tangential momentum and energy exchange of the molecules with the walls, respectively. Thermal accommodation coefficient is defined by,

\[ \sigma_T = \frac{dE_i - dE_r}{dE_i - dE_w} \]  \hspace{1cm} (2.32)

where \( dE_i \) and \( dE_r \) denote the energy fluxes of incoming and reflected molecules per unit time, respectively, and \( dE_w \) represents the energy flux if all the incoming molecules had been re-emitted with the energy flux corresponding to the surface...
temperature $T_w$. The perfect energy exchange case corresponds to $\sigma_T = 1$. Similarly, the tangential momentum coefficient can be defined for tangential momentum exchange of gas molecules with surfaces as follows:

$$\sigma_v = \frac{\tau_i - \tau_r}{\tau_i - \tau_w}$$

(2.33)

Here, $\tau_i$ and $\tau_r$ show the tangential momentum of incoming and reflected molecules, respectively, and $\tau_w$ is the tangential momentum of re-emitted molecules, corresponding to that of the surface (for stationary surfaces $\tau_w = 0$). The case of $\sigma_v = 0$ is called specular reflection, where the tangential velocity of the molecules reflected from the walls is unchanged, but the normal velocity of the molecules is reversed due to the normal momentum transfer to the wall. The case of $\sigma_v$ is called diffuse reflection. In this case the molecules are reflected from the walls with zero average tangential velocity. Therefore, the diffuse reflection is an important case for tangential momentum exchange (and thus friction) of the gas with the walls.

Eq. (2.29) is first proposed by Maxwell in 1879. The second term in Eq. (2.29) is associated with the thermal creep phenomenon, which can be important in causing pressure variation along channels in the presence of tangential temperature gradients. Eq. (2.30) is by von Smoluchowski and represents a model for temperature jump effects.

Eqs. (2.29) and (2.30) can be written after non-dimensionalization with a reference velocity and temperature as follows:

$$U_s - U_w = \frac{2 - \sigma_v}{\sigma_v} Kn \frac{\partial U_s}{\partial n} + \frac{3}{2\pi} \frac{(\gamma - 1) Kn^2 Re}{E_c} \frac{\partial T}{\partial s}$$

(2.34)
\[ T_s - T_w = \frac{2 - \sigma_T}{\sigma_T} \left[ \frac{2\gamma}{\gamma + 1} \right] \frac{Kn}{Pr} \frac{\partial T}{\partial n} \]  

and,

\[ Ec = \frac{u^3}{C_s \Delta T} \]  

where \( Ec \) is the Eckert number, \( \Delta T \) is a specified temperature difference in the domain and the capital letters are indicating non-dimensional quantities. Also, \( n \) and \( s \) denote the outward normal unit vector and tangential unit vector, respectively.
CHAPTER III
MATHEMATICAL MODEL

In this chapter, governing equations, computational schemes and algorithms, and the computational solver are presented and relevant information about the boundary condition modifications and their verification are provided.

3.1. Numerical method

CFL3D is a Reynolds-averaged Navier-Stokes flow solver on structured grids. The original version of CFL3D was developed in the early 1980's in the Computational Fluids Laboratory at NASA Langley Research Center. CFL3D solves the time-dependent conservation law form of the Reynolds-averaged Navier-Stokes equations. The spatial discretization involves a semi-discrete finite-volume approach. Upwind-biasing is used for the convective and pressure terms, while central differencing is used for the shear stress and heat transfer terms. Time advancement is implicit with the ability to solve steady or unsteady flows. Multigrid and mesh sequencing are available for convergence acceleration. Numerous turbulence models are provided, including zero-equation, one-equation, and two-equation models. Multiple-block topologies are possible with the use of one-to-one blocking, patching, overlapping, and embedding. CFL3D does not contain any grid generation software. Grids must be pre-processed and supplied external to the code. More information about the code CFL3D can be found in (Rumsey et al [78]; Baysal et al [79]; Bartels et al [80]).
3.1.1. Governing Equations

The computational code CFL3D v6.3, which is employed for the present numerical calculations, uses the three dimensional, compressible, time dependent Navier-Stokes equations. The set of equations can be written in terms of generalized coordinates, \((\xi, \eta, \zeta)\), as follows:

\[
\frac{\partial \hat{Q}}{\partial t} + \left( \frac{\partial \hat{F}}{\partial \xi} + \frac{\partial \hat{G}}{\partial \eta} + \frac{\partial \hat{H}}{\partial \zeta} \right) - \left( \frac{\partial \hat{F}}{\partial \xi} + \frac{\partial \hat{G}}{\partial \eta} + \frac{\partial \hat{H}}{\partial \zeta} \right) = 0
\] (3.1)

A general, three dimensional transformation between the Cartesian variables \((x_1, x_2, x_3)\) and generalized coordinates \((\xi, \eta, \zeta)\) can be involved by using the variable \(J\) which represents the Jacobian of the coordinate transformation:

\[
J = \frac{\partial (\xi, \eta, \zeta, t)}{\partial (x_1, x_2, x_3, t)}
\] (3.2)

In equation (3.1), \(\hat{Q}\) is the vector of variables, which are density, momentum and total energy per unit volume, and is given by

\[
\hat{Q} = \frac{Q}{J} = \frac{1}{J}
\]

\[
\begin{bmatrix}
\rho \\
\rho u_1 \\
\rho u_2 \\
\rho u_3 \\
\rho e
\end{bmatrix}
\] (3.3)

The inviscid flux terms are:

\[
\hat{F} = \frac{F}{J} = \frac{1}{J}
\]

\[
\begin{bmatrix}
\rho U_1 \\
\rho U_1 u_1 + \xi, p \\
\rho U_1 u_2 + \xi, p \\
\rho U_1 u_3 + \xi, p \\
(\rho e + p)U_1 - \xi, p
\end{bmatrix}
\] (3.4)
\begin{align*}
\dot{G} = \frac{G}{J} = \frac{1}{J} \begin{bmatrix} 
\rho U_2 \\
\rho U_2 u_1 + \eta_x p \\
\rho U_2 u_2 + \eta_x p \\
\rho U_2 u_3 + \eta_x p \\
(\rho e + p) U_2 - \eta p 
\end{bmatrix} \\
\dot{H} = \frac{H}{J} = \frac{1}{J} \begin{bmatrix} 
\rho U_3 \\
\rho U_3 u_1 + \zeta_x p \\
\rho U_3 u_2 + \zeta_x p \\
\rho U_3 u_3 + \zeta_x p \\
(\rho e + p) U_3 - \zeta p 
\end{bmatrix}
\end{align*}
\quad (3.5)

where the contravariant velocities are given as:
\begin{align*}
U_1 &= \xi_x u_1 + \xi_y u_2 + \xi_z u_3 + \xi_t \\
U_2 &= \eta_x u_1 + \eta_y u_2 + \eta_z u_3 + \eta_t \\
U_3 &= \zeta_x u_1 + \zeta_y u_2 + \zeta_z u_3 + \zeta_t
\end{align*}
\quad (3.7)

and the viscous flux terms are given as follows:
\begin{align*}
\dot{F}_v = \frac{F_v}{J} = \frac{1}{J} \begin{bmatrix} 
0 \\
\xi \tau_{xx} + \xi \tau_{xy} + \xi \tau_{xz} \\
\xi \tau_{xy} + \xi \tau_{yy} + \xi \tau_{yz} \\
\xi \tau_{xz} + \xi \tau_{yz} + \xi \tau_{zz} \\
\xi \hat{b}_x + \xi \hat{b}_y + \xi \hat{b}_z 
\end{bmatrix} \\
\dot{G}_v = \frac{G_v}{J} = \frac{1}{J} \begin{bmatrix} 
0 \\
\eta \tau_{xx} + \eta \tau_{xy} + \eta \tau_{xz} \\
\eta \tau_{xy} + \eta \tau_{yy} + \eta \tau_{yz} \\
\eta \tau_{xz} + \eta \tau_{yz} + \eta \tau_{zz} \\
\eta \hat{b}_x + \eta \hat{b}_y + \eta \hat{b}_z 
\end{bmatrix}
\end{align*}
\quad (3.8)
\quad (3.9)
The shear stress and heat flux terms are defined in tensor notations as follows:

$$
\frac{\hat{H}_v}{\hat{J}} = \frac{H_v}{J} = \frac{1}{J} \begin{bmatrix}
0 \\
\zeta_x r_{xx} + \zeta_y r_{xy} + \zeta_z r_{xz} \\
\zeta_x r_{xy} + \zeta_y r_{yy} + \zeta_z r_{yz} \\
\zeta_x r_{xz} + \zeta_y r_{yz} + \zeta_z r_{zz} \\
\zeta_x b_x + \zeta_y b_y + \zeta_z b_z
\end{bmatrix}
$$

(3.10)

Hence, the pressure is obtained by the equation of state for a perfect gas:

$$
p = \gamma - 1 \left[ e - \frac{P}{2} \left( u_1^2 + u_2^2 + u_3^2 \right) \right]
$$

(3.14)

The variables in the above equations are nondimensionalized with respect to the free stream density, $\bar{\rho}_m$, the free stream speed of sound, $\tilde{a}_m$, the free stream molecular viscosity $\tilde{\mu}_m$ and the length $L$.

For the thin layer approximations, the derivatives in $\zeta$ direction which is normal to the wall are retained in the shear stress and heat flux terms. The pressure is nondimensionalized by the term $\rho_m a_m^2$. The free stream Reynolds number is defined as,

$$
Re = \frac{\rho_m U_m L}{\mu_m}
$$

(3.15)
and the Prandtl number is given by

\[ \text{Pr} = \frac{\mu C_p}{k} \]  

(3.16)

In the present study, the Prandtl number is chosen to be 0.72. The dimensionless
viscosity is related to the temperature by Sutherland's law as:

\[ \mu = T^\frac{1+c}{T+c} \]  

(3.17)

where \( T \) is the non-dimensional temperature and \( c \) is the Sutherland's constant given
by \( c \approx 110.4/T_w \).

### 3.1.2. Numerical Algorithm

A semi discrete finite volume formulation is used in the numerical algorithm of the
solver for a consistent approximation to the conservation laws in integral form:

\[
\frac{\partial}{\partial t} \iiint_{\Omega} Q dV + \iint_{\partial \Omega} \vec{f} \cdot \vec{n} dS = 0
\]

(3.18)

Here \( \vec{f} \) represents the net flux through a surface \( \partial \Omega \) with unit outward normal \( \vec{n} \)
containing the time invariant volume \( \Omega \). Integrating Eq. (3.18) over a control volume
bounded by constant \( \xi, \eta \) and \( \zeta \) surfaces, the semi-discrete form can be obtained:

\[
\frac{\partial \hat{Q}}{\partial t}_{i,j,k} + \left( \hat{F}_v - \hat{F}_{v,j+1/2,k} \right)_{i-1/2,j,k} - \left( \hat{F}_v - \hat{F}_{v,j-1/2,k} \right)_{i+1/2,j,k} \\
+ \left( \hat{G} - \hat{G}_{v,j+1/2,k} \right)_{i,j-1/2,k} - \left( \hat{G} - \hat{G}_{v,j-1/2,k} \right)_{i,j+1/2,k} \\
+ \left( \hat{H} - \hat{H}_{v,j,k+1/2} \right)_{i,j,k-1/2} - \left( \hat{H} - \hat{H}_{v,j,k-1/2} \right)_{i,j,k+1/2} = 0
\]

(3.19)

\[ \Delta \xi = \xi_{i+1/2,j,k} - \xi_{i-1/2,j,k} \]

(3.20)

\[ \Delta \eta = \eta_{i,j+1/2,k} - \eta_{i,j-1/2,k} \]

(3.21)
\[ \Delta \zeta = \zeta_{i,j,k+1/2} - \zeta_{i,j,k-1/2} \]  
(3.22)

In equations (3.20), (3.21) and (3.22), \( \Delta \xi, \Delta \eta \) and \( \Delta \zeta \) are taken to be equal to unity in the computational domain.

The discrete value of \( \tilde{Q}_{i,j,k} \) is taken as the average value of a unit computational cell and discrete values of \( \tilde{F}, \tilde{G}, \) and \( \tilde{H} \) are regarded as face average values. Roe’s upwind flux-difference splitting technique (Hirch [82]; Hirch [83]) was used for the convective and pressure terms and it will be discussed in the next section. The shear stress and heat transfer terms are centrally differenced.

3.1.2.1. Roe’s Flux Difference Splitting Technique

In this technique of Roe, the interface flux in the \( \xi \) direction is written as

\[ \tilde{F}_{i+1/2} = \frac{1}{2} \left[ \tilde{F}(q_L) + \tilde{F}(q_R) - |\tilde{A}_{\text{inv}}|(q_R - q_L) \right]_{i+1/2} \]  
(3.23)

where \( \tilde{A}_{\text{inv}} \) is the evaluation of \( A_{\text{inv}} \) with Roe-averaged variables defined below.

\[ A = \left( \frac{\partial \tilde{F} - \tilde{F}_v}{\partial Q} \right) \]  
(3.24)

\[ |\tilde{A}_{\text{inv}}| = |A_{\text{inv}}(q)| \]  
(3.25)

\( A_{\text{inv}} \) is the inviscid part of the matrix \( A \), that is,

\[ A_{\text{inv}} = \frac{\partial \tilde{F}}{\partial Q} = T \Lambda T^{-1} = T(\Lambda^+ + \Lambda^-)T^{-1} \]  
(3.26)

\[ |A_{\text{inv}}| = |T|\Lambda|T^{-1} \]  
(3.27)
Here, $\Lambda$ is the diagonal matrix of eigenvalues of the matrix $A_{\text{inv}}$. $T$ is the matrix of right eigenvectors as columns and $T^{-1}$ is the matrix of left eigenvectors as rows. They are all evaluated using Roe-averaged values such that the term given below is satisfied exactly.

$$\hat{F}(Q_R) - \hat{F}(Q_L) = |A_{\text{inv}}|(Q_R - Q_L)$$

(3.28)

Here, the term $|A_{\text{inv}}|(Q_R - Q_L)$ can be written as below:

$$|A_{\text{inv}}|(Q_R - Q_L) = |\hat{A}_{\text{inv}}| \Delta Q =
\begin{bmatrix}
\alpha_4 \\
\tilde{u} \alpha_4 + \tilde{\xi} \alpha_5 + \alpha_6 \\
\tilde{u} \alpha_4 + \tilde{\xi} \alpha_5 + \alpha_7 \\
\tilde{u} \alpha_4 + \tilde{\xi} \alpha_5 + \alpha_8 \\
\tilde{H} \alpha_4 + (\tilde{\xi} - \tilde{\eta}) \alpha_5 + \tilde{u} \alpha_6 + \tilde{v} \alpha_7 + \tilde{w} \alpha_8 - \frac{\tilde{a}^2 \alpha_1}{\gamma - 1}
\end{bmatrix}$$

(3.29)

where,

$$\alpha_1 = \left[ \frac{\nabla \tilde{\xi}}{J} \right] \left( \Delta \rho - \frac{\Delta p}{\tilde{a}^2} \right)$$

$$\alpha_2 = \frac{1}{2\tilde{a}^2} \left[ \frac{\nabla \tilde{\xi}}{J} \right] \tilde{U} + \tilde{a} \left( \Delta \rho + \tilde{p} \tilde{a} \Delta \tilde{U} \right)$$

$$\alpha_3 = \frac{1}{2\tilde{a}^2} \left[ \frac{\nabla \tilde{\xi}}{J} \right] \tilde{U} - \tilde{a} \left( \Delta \rho - \tilde{p} \tilde{a} \Delta \tilde{U} \right)$$

$$\alpha_4 = \alpha_1 + \alpha_2 + \alpha_3$$

$$\alpha_5 = \tilde{a} (\alpha_2 - \alpha_3)$$

$$\alpha_6 = \left[ \frac{\nabla \tilde{\xi}}{J} \right] \tilde{U} \left( \tilde{\rho} \Delta u - \tilde{\xi}, \tilde{\rho} \Delta \tilde{U} \right)$$

$$\alpha_7 = \left[ \frac{\nabla \tilde{\xi}}{J} \right] \tilde{U} \left( \tilde{\rho} \Delta u - \tilde{\xi}, \tilde{\rho} \Delta \tilde{U} \right)$$

$$\alpha_8 = \left[ \frac{\nabla \tilde{\xi}}{J} \right] \tilde{U} \left( \tilde{\rho} \Delta u - \tilde{\xi}, \tilde{\rho} \Delta \tilde{U} \right)$$

(3.30)

The tilde (\text{~}) symbol indicates the following Roe-averaged variables.
\[ \tilde{\rho} = \sqrt{\rho_L \rho_R} \]
\[ \tilde{u} = \frac{u_L + u_R \sqrt{\rho_R / \rho_L}}{1 + \sqrt{\rho_R / \rho_L}} \]
\[ \tilde{v} = \frac{v_L + v_R \sqrt{\rho_R / \rho_L}}{1 + \sqrt{\rho_R / \rho_L}} \]
\[ \tilde{w} = \frac{w_L + w_R \sqrt{\rho_R / \rho_L}}{1 + \sqrt{\rho_R / \rho_L}} \]
\[ \tilde{H} = \frac{H_L + H_R \sqrt{\rho_R / \rho_L}}{1 + \sqrt{\rho_R / \rho_L}} \]
\[ \tilde{a}^2 = (y - 1) \tilde{H} - \frac{\tilde{u}^2 + \tilde{v}^2 + \tilde{w}^2}{2} \]

and,
\[ \tilde{U} = \frac{1}{|\nabla \xi|} \left( \xi_x \tilde{u} + \xi_y \tilde{v} + \xi_z \tilde{w} + \xi_t \right) \]

3.1.2.2. Moving Deforming Mesh

CFL3D has the capability to perform computations for prescribed surface motion in two ways. One of them is the prescribed (or user specified) rigid grid motion. In this mode, the entire grid or set of grids translates or rotates in a manner prescribed by user input. The other one is the prescribed surface motion with deforming mesh. In this mode, the surface(s) prescribed by the user translate or rotate and the mesh deforms accordingly. These types of motion are available only when the code is running in unsteady mode. In the present study the prescribed surface motion with deforming mesh is used to accurately model the synthetic jets membrane motion.

The mesh is deformed in two steps. The first step moves control points, also called sub-grid or node points. This first step can be performed by simply using an
exponential decay function that transmits surface motion into the flow field in an exponentially decaying manner via control points. This approach is computationally fast, and robust for small motion. It does not however retain grid orthogonality as a surface rotates. Alternatively, this first step can be performed using the Finite Macro-Element method. In this approach, sub-grid points are the node points of a finite element set. These sets of points are solved using a fictitious material property that produces very stiff (in fact essentially rigid) material near a moving surface and relatively pliable material away from a surface. This approach is more computationally intensive, but produces relatively orthogonal grids as a surface rotates. The second step in mesh deformation is composed of line transfinite interpolation (TFI) between sub-grid points, surface TFI of sub-grid faces composed of four adjacent control points and volume TFI of sub-blocks composed of eight adjacent control points. The end result of this final step is the movement of all mesh points in the grid.

The surface velocities are determined as follows:

\[
\begin{align*}
    u_w &= M_w C_q \xi_z / \left| \triangledown \xi_z \right| \frac{1}{\gamma P_b} + u_{mesh} \\
    v_w &= M_w C_q \xi_y / \left| \triangledown \xi_y \right| \frac{1}{\gamma P_b} + v_{mesh} \\
    w_w &= M_w C_q \xi_x / \left| \triangledown \xi_x \right| \frac{1}{\gamma P_b} + w_{mesh}
\end{align*}
\]  \hspace{1cm} (3.33)

In these equations, \( u_{mesh}, v_{mesh}, w_{mesh} \) are the velocity components of the computational domain.
3.1.3. Initial and Boundary Conditions

In order to solve the equations of motion, boundary conditions are required to be set on all sides of the domain as well as the physical surfaces of any objects lying in the domain. In other words, boundary conditions must be applied at each face of the computational block.

There are two types of boundary condition representations employed in CFL3D, namely, cell-center and cell-face. For cell-center type boundary conditions, the flow field variables are specified at *ghost* points corresponding to two cell center locations analytically extended outside of the grid (Fig. 3.1). For cell-face type boundary conditions, the flow field variables and their gradients are specified at the cell face boundary (Fig. 3.2). For the computations carried out throughout the present thesis, the boundary conditions implemented are both cell-center and cell-face type boundary conditions. In Chapter IV, the computational domains for the problems in hand are presented with the specified boundary conditions.

![Figure 3.1. Cell-center type boundary conditions.](image)

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The present study is three fold. First, the geometric and actuation parameters of a synthetic jet actuator are investigated (Section 5.1). The synthetic jet exhausting into a quiescent external medium is modeled using extrapolation boundary condition, viscous surface boundary condition, and symmetry plane boundary condition presented in CFL3DV6.3. Second, we discuss the separation control in a microchannel with a backward facing step using a synthetic jet actuator (Section 5.2). The channel and synthetic jet actuator are simulated using freestream boundary condition at the inlet, specified pressure ratio boundary condition at the outlet and viscous surface boundary condition on the walls. Finally, Section 5.3 talks about the effectiveness of the synthetic jet actuators for thermal management of a heat-dissipating microelectronic device placed in a channel. The inlet and the outlet are modeled using a specified subsonic inflow boundary condition and specified pressure ratio boundary condition, respectively. On the surfaces, viscous surface boundary condition is imposed.

The freestream is modeled by cell-center type of boundary conditions. The five flow field variables are set equal to the initial values which are,
\[ \begin{align*}
    \rho_{\text{initial}} &= 1.0 \\
    u_{\text{initial}} &= M_\infty \cos \alpha \cos \beta \\
    v_{\text{initial}} &= -M_\infty \sin \beta \\
    w_{\text{initial}} &= M_\infty \sin \alpha \cos \beta \\
    p_{\text{initial}} &= \rho_{\text{initial}} (a_{\text{initial}})^2 / \gamma
\end{align*} \] (3.33)

where \( a_{\text{initial}} = 1.0 \). The freestream incidence values of \( \alpha = 0 \) and \( \beta = 0 \) are imposed for the present study.

The \textit{symmetry} is modeled using cell-center type boundary conditions across a specified axis. The ghost point density values are set to their mirror image counterparts. For \textit{extrapolation}, cell-center type boundary conditions are used and the ghost points are extrapolated from the computational domain.

The \textit{specified pressure ratio} is generally imposed to model the outflows for internal flows and is of cell-center type. A single pressure ratio \(( \tilde{p} / \tilde{p}_\infty )\) is specified on input. This pressure ratio is used to set both cell-center pressure boundary values. Extrapolation from inside the computational domain is used to set the boundary values for \( \rho, u, v \) and \( w \).

The \textit{subsonic inflow} is used for inflow into channel where flow variables are known and it can be specified in different ways. In the present study, one type of inflow boundary condition is implemented where the user gives the density and velocity components, and the pressure is extrapolated from the interior.

The \textit{viscous surfaces} are applied in the present study on the walls of the domains to enforce the no-slip condition as well as the slip boundary conditions. Viscous boundary conditions are of cell-face type. Two pieces of auxiliary information are supplied on input: the wall temperature \(( \tilde{T}_w / \tilde{T}_\infty )\) and the mass flow \(( C_q )\) where \( C_q = (\rho u_{\text{normal}}) / (\rho u)_\infty \) \(( C_q \) is zero if there is no flow through the wall).
The no-slip ($u_1 = u_2 = u_3 = 0$) conditions are applied at the body surfaces where the Knudsen number based on the characteristic length corresponds to the continuum flow regime. On the other hand, for the test cases which has a characteristic length at micron level, slip boundary conditions are implemented on the walls and this option has to be added to the code CFL3D.

Surface boundary condition routines of the code CFL3D are modified as follows. Usual no-slip and temperature wall boundary conditions (BC) are augmented to account for the slip flow and temperature jump boundary conditions encountered in MEMS devices. The first-order Maxwell-Smoluchowski slip-boundary conditions (Gad-el-Hak [3]; Beskok [17] and Agarwal and Yun [76]) in Cartesian coordinates are:

\[
U_s - U_w = \frac{2 - \sigma_s}{\sigma_v} \frac{2}{\rho} \sqrt{\frac{\pi}{8RT_w}} \tau_s + \frac{3 \Pr (\gamma - 1)}{4} \frac{\gamma R \rho T_w}{\gamma + 1} (-q_s)
\] (3.34)

\[
T_s - T_w = \frac{2 - \sigma_T}{\sigma_T} \frac{(\gamma - 1)}{\gamma + 1} \frac{4}{\rho R} \sqrt{\frac{\pi}{8RT_w}} (-q_n)
\] (3.35)

In these equations, subscript $s$ denotes the slip flow variables on the solid surface of the body; $\gamma$ is the ratio of specific heats, $\rho$ is the density and $R$ is the specific gas constant; $q_n$ and $q_s$ are the normal and tangential heat-flux components and $\tau_s$ is the shear stress component pertaining to the skin friction; $U_w$ and $T_w$ are the reference wall velocity and temperature, respectively; $Pr$ is the Prandtl number. $\sigma_v$ and $\sigma_T$ are the tangential momentum and energy "accommodation" coefficients. In the present study, full diffuse reflection is assumed both for the tangential momentum and energy exchange ($\sigma_v = \sigma_T = 1$). The slip and jump boundary conditions given above are first-
order in Knudsen number. The application of boundary condition formulations requires the variation of tangential velocity in the normal direction to the wall. Also coded and used herein is a second-order extension of the Maxwell’s slip velocity boundary condition (Beskok et al [14]):

\[
U_s - U_w = \frac{2 - \sigma_s}{\sigma_v} \left[ \frac{Kn}{1 - bKn} \left( \frac{\partial U}{\partial n} \right)_s \right]
\]  

(3.36)

Where \( b \) is the general slip coefficient determined analytically in the slip and early transition flow regimes. For the present examples, the value of \( b=1 \) is employed and the thermal creep term has been omitted.

In addition to the slip boundary conditions, the surface boundary condition routines of the code CFL3D are modified in order to implement a constant heat flux boundary condition to prescribe the wall temperature \( (\bar{T}_w/\bar{T}_w) \) on the relevant walls. This is done simply by taking advantage of the adiabatic wall option which is already coded in CFL3D.

### 3.2. Verification of Modified Boundary Conditions

CFL3D is coded to solve the three-dimensional time-dependent, compressible Navier-Stokes equations subject to various boundary conditions and has been modified as part of this study to account for the slip velocity and temperature jump conditions encountered in MEMS geometries for the \( Kn \) numbers up to 0.1. These modifications have been validated for simpler geometries (Beskok [17]).

In order to verify the computational model and the boundary condition modifications, three benchmark cases are considered: Flow in a straight
microchannel, flow past a micro backward facing step, and flow through a micro filter.

### 3.2.1. Straight Micro Channel

Flow through a two-dimensional, isothermal, subsonic microchannel with a channel length-to-height ratio of 20 ($L/h$) is considered (Fig. 3.3). Knudsen number at channel outlet is $Kn_{out}=0.2$ while at inlet it is $Kn_{in}=0.088$. Inlet-to-outlet pressure ratio is 2.28. The reference Mach ($M$) and Reynolds ($Re$) numbers are 0.0725 and 1.22, respectively, while 273 °K is taken as the reference temperature. This flow is laminar and can be considered to be in the slip flow regime, i.e. $Kn \approx 0.1$. The working fluid is the diatomic nitrogen. This geometry has also been studied by Beskok [17] using both a DSMC solver and a spectral-element-based continuum CFD solver, and by Agarwal and Yun [76] using a higher order fluid dynamics model of Burnett equations.

![Figure 3.3](image)

**Figure 3.3.** Computational domain for straight channel: every 3rd grid point is shown.

In the present computations, both the first order (Eq. 3.34) and the second order (Eq. 3.36) velocity slip boundary formulations have been employed. To establish the grid dependence, the computations have been performed using two different meshes with 101*25 and 201*49 cells. Both grids have produced very similar results.
Excellent agreement has been obtained with the analytical solutions available for the centerline pressure distribution through the channel (Beskok et al. [14]). The effect of compressibility, even at such low speeds, has also been observed. The pressure distribution is nonlinear and slightly lower in magnitude compared to the no-slip solution. As seen in Figure 3.4, the centerline pressure distribution is not deemed sensitive to the order of accuracy of the slip wall formulation and there is at least an order of magnitude difference between the no slip and slip cases. In Figure 3.4, the deviation of the computed centerline pressure from the analytically obtained pressure, \( \Delta(p/p_{\text{out}}) \), is plotted:

\[
\Delta(p/p_{\text{out}}) = (p/p_{\text{out}})_{\text{analytical}} - (p/p_{\text{out}})_{\text{computational model}}
\]

\[ (3.37) \]

where \( p_{\text{out}} \) is the exit channel pressure used to normalize the centerline pressure values.

**Figure 3.4.** Deviation of centerline pressure (Eq. 3.37) distribution through the micro duct for different computational models, \( Kn_{\text{in}} = 0.088 \).
The computed wall slip velocity distributions, in comparison with the results of Beskok [17] and Agarwal and Yun [76] and the analytical solution, are presented in Figure 3.5. The results with the first order slip boundary condition match exactly the analytical results, which are based on first order approximations as well. The second order results match the Navier Stokes and Burnett equation solutions of Agarwal and Yun [76] perfectly. The DSMC results of Beskok [17] are slightly different for the first half of the channel but match the other results towards the exit of the duct. The increase in the mass flow due to the wall slip is about 13% as also predicted by Agarwal and Yun [76]. In conclusion, the present implementation of both the first and the second order slip boundary condition formulations has been deemed appropriate for microflows.

![Figure 3.5. Variation of slip velocity along the microduct wall, Kn_{ln} = 0.088.](image-url)
3.2.2. Micro Filter

Another validation case considered for the present study is a flow through a micro-filter. Analysis of gas flows through micro-filters requires the consideration of three fundamental issues: rarefaction, compressibility and geometric complexity (Ahmed and Beskok [84]). The rarefaction is due to the small characteristic length scales of micro-filters \( L \), which are comparable to the local mean free path \( \lambda \). Compressibility effects are important when there are large density variations in the micro fluidic system, particularly, when there are pressure and/or temperature fluctuations. In their simplest form, a micro-filter is a very short channel or sudden constriction. A schematic view and characteristic dimensions of a section of a rectangular micro-filter array are presented in Figure 3.6. In Figure 3.7, the numerical grid used for the computations is shown.

![Figure 3.6. Cross-sectional view and the characteristics dimensions of the micro filter.](image-url)

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Experimental and numerical studies have shown that the flow in the micro-filters strongly depends on the opening factor $\beta$: the ratio of the hole-height ($h$) to the total filter length ($L$). Considering that the filter holes repeat in a periodic fashion and the symmetry between the two periodic sides, gas flow through only one-half hole is computed by imposing the symmetry boundary conditions in the streamwise direction. In the present study, the ratio of the height to the length at the hole opening is $h/l=1.5$, with an opening factor $\beta = h/L = 0.6$, where $h=1.2$, $l=0.8$ and $L=2$ microns. The gray shaded areas correspond to physical surfaces of the micro-filter, where fully accommodating, diffuse reflection boundary conditions are applied. The surface temperature is kept at 300 K. The reference length scale used in the definition of $Kn$ and $Re$ is the hole-height $h$ (Fig. 3.6).

The computational domain extends about 11 filter lengths upstream and about 20 filter lengths downstream. The grid consists of three blocks (normal*streamwise
cells): upstream region (33*73), filter region (17*33) and the downstream region (33*81). The predictions are obtained for two flow cases using the same filter dimensions given in Table I.

Table 3.1. Case definitions for the micro filter.

<table>
<thead>
<tr>
<th>Case</th>
<th>$M_{in}$</th>
<th>$M_{out}$</th>
<th>$Re$</th>
<th>$Kn_{in}$</th>
<th>$Kn_{out}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>0.17</td>
<td>0.23</td>
<td>7.51</td>
<td>0.054</td>
<td>0.071</td>
</tr>
<tr>
<td>II</td>
<td>0.21</td>
<td>0.28</td>
<td>8.82</td>
<td>0.036</td>
<td>0.047</td>
</tr>
</tbody>
</table>

The predictions from Case I (Table 3.1) are compared with the computational results of Ahmed and Beskok [84], where the geometry differs from the present one only with its rounded corners. The streamwise velocity and temperature values, normalized with corresponding inflow values along the centerline of the filter, are in good agreement with those of Ahmed and Beskok [84] (Fig. 3.8). The flow which is uniform up to $y/L=3.3$, starts to develop before it reaches the filter inlet located $y/L=4.3$.

The rarefaction effects are investigated by comparing the results of two different outlet $Kn$ simulations. Rarefaction is known to cause skin friction reduction. In Figure 3.9, the shear stress distribution normalized with inlet dynamic pressure is presented on the filter surface as a function of the surface length for both Cases I and II. In this figure, the non-dimensional values of distance at 0, 0.33, 1, and 1.33, correspond to bottom first point, first corner, second corner and top last point along the filter surface, respectively. The shear stress values are lower for the higher $Kn$ flow of Case I (top figure). Comparisons of the slip with no-slip cases reveal that shear stresses decrease as a result of the velocity slip on the walls. In Figure 3.9, sign

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
reversal in the shear stress indicates local flow separation. The recirculation region is larger in the case of larger $Re$ flow of Case II (bottom figure).

![Graphs showing normalized streamwise velocity and temperature variations along the centerline of micro filter.]

**Figure 3.8.** a) Normalized streamwise velocity variations along the centerline of micro filter, b) normalized streamwise temperature variations along the centerline of micro filter.
Figure 3.9. Variation of shear stress along micro filter wall. Case I (top) and Case II (bottom).
3.2.3. Backward Facing Step

Flow past a backward facing step is computed to study the effect of slip boundary conditions on a separated flow (Fig. 3.10). The outlet channel height $h_c$ is 1.25 $\mu$m with a ratio of channel length-to-exit height of 5.6. The entry to the channel is also simulated. The channel inlet is located at $x/h_c=0.86$ and the step height $s$ is taken as $s/h_c=0.467$ (Fig. 3.11). The first and second order slip boundary conditions are employed. Simulation is performed for inlet Mach number $M=0.47$ and inlet $Re$ number $Re=80$. The outlet $Kn$ number is 0.018. Sample results are presented for an inlet-to-outlet pressure ratio of $P_{in}/P_{out}=2.32$ and inlet temperature is 330 K. The walls are kept at a constant temperature of 300 K. The working fluid is nitrogen.

![Figure 3.10. Separated flow in a micro backward facing step.](image)

The computational grid consists of two domains. The grid before the step (section I) is 49*17, and the domain after (section II) step has a grid of 161*33 cells (Fig. 3.11). To test the grid independency, a finer grid of 97*33 cells (section I) and 241*65 cells (section II) have also been used. Results from both grids were identical.
up to the fifth digit. The mass flow is monitored for convergence until a constant mass flow is achieved throughout the channel.

![Diagram](image_url)

**Figure 3.11.** Micro backward facing step with its numerical grid.

For this case, the streamwise pressure variation, normalized with the inlet dynamic head, \( q_{in} = 0.5 \rho_{in} U_{in}^2 \), and the streamwise velocity, normalized with the local speed of sound \( a \), at about the center of entrance \( y/h_c = 0.755 \) are shown in Figure 3.12 and Figure 3.13, respectively. The results compare well with the DSMC computations of Beskok [17], where the results are for a slightly different flow condition of \( Kn_{out} = 0.04 \).

![Graph](image_url)

**Figure 3.12.** Streamwise variation of pressure.

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 3.13. Variation of normalized streamwise component of velocity.
CHAPTER IV
RESULTS and DISCUSSION

4.1. Modeling and Characterization of Synthetic Jets

As the first part of the present study, the effectiveness of a two-dimensional synthetic jet is studied using numerical simulations. The computations are carried out for the synthetic jet emerging into a quiescent external medium. The primary focus is on the analysis of the design space determined by the geometric and flow-type design variables that identify the effectiveness of the synthetic jet in terms of the jet velocity and vortex dynamics.

Consider a synthetic jet actuator exhausting into a quiescent external flow as shown in Figure 4.1. The actuator comprises of a cavity and a throat connecting the flow inside the cavity to the external flow. In a synthetic jet, the actuating flow is generated at the orifice by oscillating a membrane opposite to the orifice. The membrane oscillation is characterized by the maximum membrane oscillation amplitude, \( A \), and membrane oscillation frequency, \( \omega \). The cavity is defined by the cavity height, \( H \), and the cavity width, \( W \). The cavity geometry can be of any shape; a triangular shaped geometry is illustrated in Figure 4.1. \( d_o \) is the throat width and \( h_o \) is the throat height which together characterize the throat geometry. Fluctuating jet flow with a jet velocity, \( V_{jet} \), produced by the oscillatory motion of the membrane, interacts with the main flow and transfers linear momentum to the external flow. The working fluid is nitrogen and it is described by its dynamic viscosity, \( \mu \), and density, \( \rho \).
Many studies have previously investigated the synthetic jets in order to provide useful knowledge about the physics of the jet flow. These studies employed either a velocity distribution at the orifice exit plane excluding the cavity (Kral et al [6]; Mallinson et al [7]), or a pressure boundary condition (Rizetta et al [38]), a moving piston (Lee and Goldstein [39]), or a moving membrane (Aslan et al [1]; Mittal et al [8]) at the bottom side of the cavity. Only the moving membrane accurately represents the physical situation and the other methods were simplification of the problem.

In this study, the membrane of the actuator is modeled in a realistic manner as a moving boundary to accurately compute the flow inside the actuator cavity. It is assumed that the movement of the membrane is only in the vertical direction and the position and shape of the membrane is formulated using the equation below,

$$ y(x,t) = \frac{1}{2} \left( A + A \sin \left( \frac{x + W/4}{W/2} \pi \right) \right) \sin(\omega t) $$

where $t$ denotes time. Modeled by Eq. (4.1), the membrane is clamped at its edges as shown in Figure 4.2.
As the membrane oscillates, the fluid is expelled from and entrained into the cavity in a periodic behavior. Under certain operating conditions, the fluid separates forming a vortex pair at the edge of the throat exit in the expulsion phase. This attribute is termed as vortex formation. If the vortex pair moves outwards, away from the throat exit under its own induced velocity; then it is defined as vortex shedding.

The synthetic jet flow can be described by two independent dimensionless parameters depending on which a vortex formation and/or vortex shedding may occur. Although many choices are possible, the important parameters may be the relative importance of the viscous effects and the relative importance of the unsteady effects. The Re number based on the jet velocity and the throat width describes the viscous effects:

\[
Re = \frac{\bar{V}_{jet} d_o}{\nu}
\]  

(4.2)

Where \( \bar{V}_{jet} \) is the time- and spatial-averaged velocity that is calculated during the expulsion phase of one membrane oscillation cycle and \( \nu \) is the kinematic viscosity.

And the unsteady effects are described by a Stokes (St) number based on the throat width and the membrane oscillation frequency as follows:

\[
St = \sqrt{\frac{\omega d_o^2}{\nu}}
\]  

(4.3)
If the $St$ number is large, the jet flow within the throat is not strongly influenced by viscous effects, whereas the flow is strongly viscous for small $St$ numbers.

The $St$ number can be related to $Re$ number by using another dimensionless parameter, namely, Strouhal ($Str$) number:

$$Str = \frac{St^2}{Re} = \frac{\omega d_o}{V_{jet}}$$  \hspace{2cm} (4.4)

Another dimensionless parameter that can define the jet formation, namely dimensionless stroke length, has been proposed by many researchers. Stroke length, $L_0$, is defined as the length of the slug of fluid pushed from the throat during the expulsion phase (Smith and Glezer [9]) and is expressed in its non-dimensional form as follows:

$$\frac{L_0}{d_o} = \frac{1}{d_o} \int_0^{T/2} u(t) dt$$  \hspace{2cm} (4.5)

Where $T$ is the oscillation period, and $u(t)$ is the centerline velocity at the throat exit. Based on a given cavity and membrane geometry, conservation of mass with the assumption of incompressible jet flow yields:

$$d_o \cdot u(t) = \int_{-d_o/2}^{d_o/2} V_{jet}(x,t) dx = \int_{-w/2}^{w/2} \frac{\partial y(x,t)}{\partial t} dx = \int_{-w/2}^{w/2} \frac{\pi}{T} \cos(2\pi ft) \sin \left( \frac{2x}{w} \right) dx$$  \hspace{2cm} (4.6)

Here, $f$ is the forcing frequency of the membrane. Thus, the spatially averaged expulsion velocity is obtained as:

$$u(t) = \frac{\pi f A W}{d_o} \cos(2\pi ft)$$  \hspace{2cm} (4.7)

The time- and spatial-averaged velocity calculated during the expulsion phase, $\overline{V_{jet}}$, is:
As can be seen from Eq. 4.8, \( \bar{V}_{jet} \) is a function of the membrane oscillation parameters as well as the geometric parameters of the actuator, which in turn defines the performance of the synthetic jet.

First, a baseline case is selected in order to observe the flow phenomenon inside the cavity and the jet flow from the orifice of the synthetic jet actuator. The results are compared with that of Mittal et al. [8]. Then, the effect of the geometrical and actuation parameters of the synthetic jet actuator has been studied on the jet velocity profile and vortex formation.

### 4.1.1 Computational Aspects

The computational domains for the synthetic jet with rectangular-shaped cavity as well as triangular-shaped one are shown in Figure 4.3. Since the computations in this section are carried out for the quiescent external flow case, the flow is symmetric. Thus only the half domain is considered. In both configurations, the flow domain consists mainly of three blocks: Cavity region, throat region and external region. The external region extends from the orifice to \( 25d_o \) in vertical direction and \( 20d_o \) in horizontal direction.

Boundary conditions on the left side of the domain and on the top boundary of the domain are prescribed as extrapolation boundary condition so that the direction of the flow will be dependent on the jet flow. The right side of the domain is the symmetry plane across which symmetry is assumed. On the solid surfaces no-slip boundary condition, unless otherwise noted, is imposed.
Figure 4.3. Computational domain and boundary conditions; a) Rectangular-shaped cavity, b) Triangular-shaped cavity.

Shown in Figures 4.4a and 4.4b are an instant of the dynamic numerical grids for rectangular-shaped and triangular-shaped cavity configurations, respectively. The numerical grids are created by Gridgen V14 [87]. The cavity regions have a grid size of 209*81 (horizontal * vertical); the orifice regions have a grid size of 17*33 while the outside domains have a grid size of 209*369.

The grid is uniform in the throat region. In the cavity region and the external region, the grids are stretched in both horizontal and vertical directions in order to save grid points in the numerical domain for the benefit of computational cost. The stretching is one way and the stretching ratio is determined in accordance with the grid spacing in the orifice region. In the quiescent external flow, the jet flow emerging from the orifice of
the synthetic jet travels away in the external domain in the y direction. Also, the flow features near the throat area are essential and have to be captured accurately. Hence, the stretching is towards the throat in both x and y directions.

Figure 4.4. An instant of numerical grid for jet configuration; a) Rectangular-shaped cavity, b) Triangular-shaped cavity (one of every 5th grid is shown).

Computational cases for the synthetic jet with quiescent external flow will be discussed in the relevant subsections. All the parameters of the synthetic jet are fixed, unless otherwise noted, at their baseline values as follows:

<table>
<thead>
<tr>
<th>$W/d_o$</th>
<th>$H/d_o$</th>
<th>$h_o/d_o$</th>
<th>$A/d_o$</th>
<th>$f$, kHz</th>
</tr>
</thead>
<tbody>
<tr>
<td>20.</td>
<td>4.</td>
<td>1.</td>
<td>0.4</td>
<td>1.</td>
</tr>
</tbody>
</table>

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Also, the velocity profiles at the throat exit will be plotted for the computational cases in the following subsections. The probe location line along which these velocity profiles are extracted is shown in Figure 4.5.

![Figure 4.5. Probe location line along which velocity profiles are extracted; Left: Rectangular-shaped cavity, right: Triangular-shaped cavity.](image)

### 4.1.2. Synthetic Jet Modeling

The synthetic jet is investigated using a Navier-Stokes solver for moving and deforming meshes. The membrane of the synthetic jet cavity is modeled in a realistic manner as a moving boundary to accurately compute the flow inside the cavity. The characteristic length is taken as the orifice width, $d_o$, which equals to $5 \times 10^{-4}$ μm. The synthetic jet geometry used is studied previously (Mittal et al [8]) and the results are compared with that of Mittal et al [8] for the verification purpose.

In Figure 4.6, time evolution of the flow field generated by the synthetic jet actuator is illustrated via vorticity contour plots in the streamwise direction. These snapshots correspond to the instants at the first four membrane cycle of the synthetic jet actuator.
Figure 4.6. Time evolution of flow field generated by synthetic jet actuator operating at a frequency of 1 kHz and amplitude of 200 μm.

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The synthetic jet flow reaches an intermittent behavior after four membrane oscillation cycles. The streamwise vorticity contour plots corresponding to the four stages of 7th membrane oscillation cycle are shown in Figure 4.7. The maximum expulsion (Max.Exp.) corresponds to the case where the membrane is at level position and moving with maximum vertical velocity towards the cavity. The minimum volume (Min.Vol.) corresponds to the instant when membrane is in its highest position. The maximum ingestion (Max.Ing.) is the level position that occurs following the minimum volume case. The membrane is at its lowest level for the Maximum Volume (Max.Vol.) stage. Three vortex pairs are present in the computational domain at all times, a new one is coming out of the orifice while an old one is leaving the domain.

As seen from the vorticity contour plots, a boundary layer forms on the walls of the synthetic jet throat. When the membrane is at the maximum expulsion stage, the boundary layer begins to separate forming a vortex attached to the throat exit. At the minimum volume stage, this vortex grows larger in size and breaks away from the throat exit moving in the positive $y$ direction. This is due to the vortex’s self-induced velocity. By the time the maximum ingestion phase begins, the vortex is ~$2.2d_o$ away from the throat exit. Vortex structure is observed not to be influenced much by the jet flow which is directed into the cavity during the ingestion stage and it continues to travel in the vertical direction. When the membrane is at its lowest level, the vortex is $5d_o$ away from the throat. It is observed that the vortex structures begin to lose their strength due to the viscous effects once they emerged from the orifice.
Figure 4.7. Rectangular cavity: Plot of vorticity contours at the four stages of membrane oscillation cycle: a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.

The dimensional streamwise velocity at the throat exit plane for the baseline case is shown in Figure 4.8 versus membrane oscillation cycle. The forthcoming quiescent external flow cases in the following sections exhibit similar behavior.

The velocity profiles at the orifice exit in the $x$ and $y$ directions are shown in Figure 4.9. The velocities are normalized with the speed of sound calculated based on the ambient temperature. As seen in Figure 4.9, the streamwise velocity reflects more of a jet-like profile during the maximum expulsion stage. The throat height used in this
Figure 4.8. Streamwise velocity at the throat exit plane vs. membrane oscillation cycle.

The simulation is not large enough to allow a significant boundary layer to develop within the throat. As a result, the \( v \)-velocity profile is far from parabolic at the maximum expulsion stage. During the maximum ingestion stage, the streamwise velocity profile is plug-like and more “horned” at the edges with peaks around \( x = -0.32d_o \) and there is a substantial velocity component in the spanwise direction. Jet flow enters the throat at maximum ingestion stage; consequently the streamwise velocity profile is similar to the entrance flow in a channel.

Figure 4.9. Velocity profiles at the throat exit at four stages of one membrane oscillation cycle; a) \( u \)-velocities, b) \( v \)-velocities.
The results obtained here are observed to be in good agreement with those of Mittal et al. [8]. The vorticity contour plots of Mittal et al. [8] at four stages are shown in Figure 4.10.

Also, the streamwise velocity profiles compared with that of Mittal et al. [8] are shown in Figure 4.11 for validation purpose. In order to give a better comparison, the streamwise velocities of our baseline case are normalized with the so-called maximum inviscid velocity, $V_{\text{max}}^{\text{inv}}$, which they used in their study:

$$V_{\text{max}}^{\text{inv}} = \frac{\pi \, A \, W \, f}{d_0}$$

where, $W$, $d_0$, $A$, $f$ are the cavity width, orifice width, membrane oscillation amplitude and membrane oscillation frequency, respectively.

As it can be seen from the plots, the streamwise velocity profile characteristics as well as the values of the streamwise velocities are matching.

Figure 4.10. Plot of vorticity contours at the four stages of membrane oscillation cycle: a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume (Mittal et al. [8]).

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
4.1.3. Synthetic Jet Design Considerations

The geometric and actuation parameters of the synthetic jet actuator are studied in terms of how they affect the jet velocity and the vortex dynamics. First, two different geometries of the synthetic jet cavity are studied for the quiescent external flow case. Then, the effectiveness of the selected synthetic jet parameters, cavity shape, orifice width, $d_o$, orifice height, $h_o$, and the membrane actuation parameters, membrane oscillation amplitude, $A$, membrane forcing frequency, $f_o$, is investigated in terms of spanwise and streamwise velocity profiles extracted from the orifice exit as well as the vortex dynamics. The computational cases are shown in Table 4.1. Case 1 refers to the baseline case with a rectangular-shaped cavity.
Table 4.1. Computational cases and synthetic jet parameters for quiescent external flow case; R: rectangular, T: triangular.

<table>
<thead>
<tr>
<th>Case</th>
<th>Cavity shape</th>
<th>$d_o/d_{ref}$</th>
<th>$h_o/d_o$</th>
<th>$W/d_o$</th>
<th>$H/d_o$</th>
<th>$A/d_o$</th>
<th>$f$, Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>R</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>2</td>
<td>T</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>3</td>
<td>T</td>
<td>1.</td>
<td>0.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>4</td>
<td>T</td>
<td>1.</td>
<td>0.5</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>5</td>
<td>T</td>
<td>1.</td>
<td>1.5</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>6</td>
<td>T</td>
<td>1.</td>
<td>2.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>7</td>
<td>T</td>
<td>1.</td>
<td>2.5</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>8</td>
<td>T</td>
<td>1.</td>
<td>3.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>9</td>
<td>T</td>
<td>0.2</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>10</td>
<td>T</td>
<td>2.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>11</td>
<td>T</td>
<td>5.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1000.</td>
</tr>
<tr>
<td>12</td>
<td>R</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.05</td>
<td>1000.</td>
</tr>
<tr>
<td>13</td>
<td>R</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.1</td>
<td>1000.</td>
</tr>
<tr>
<td>14</td>
<td>R</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.2</td>
<td>1000.</td>
</tr>
<tr>
<td>15</td>
<td>R</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.6</td>
<td>1000.</td>
</tr>
<tr>
<td>16</td>
<td>R</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.8</td>
<td>1000.</td>
</tr>
<tr>
<td>17</td>
<td>T</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>250.</td>
</tr>
<tr>
<td>18</td>
<td>T</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>500.</td>
</tr>
<tr>
<td>19</td>
<td>T</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>750.</td>
</tr>
<tr>
<td>20</td>
<td>T</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1250.</td>
</tr>
<tr>
<td>21</td>
<td>T</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1500.</td>
</tr>
<tr>
<td>22</td>
<td>T</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>1750.</td>
</tr>
<tr>
<td>23</td>
<td>T</td>
<td>1.</td>
<td>1.</td>
<td>20.</td>
<td>4.</td>
<td>0.4</td>
<td>2000.</td>
</tr>
</tbody>
</table>

4.1.3.1. Cavity Shape

A rectangular-shaped and a triangular-shaped cavity are considered in order to analyze the effect of the cavity shape on the performance of the synthetic jet actuator. The streamwise vorticity contour plots for the four membrane stages of Case 1 (rectangular-shaped cavity) were shown in Figure 4.7. Figure 4.12 shows the corresponding streamwise vorticity contour plots for Case 2 (triangular-shaped cavity).
Figure 4.12. Case 2: Plot of vorticity contours at the four stages of membrane oscillation cycle: a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.

Cavity shape is found to have considerable effect on the jet velocity at the throat exit as well as the vortex dynamics. Four vortex pairs are present for the triangular-shaped cavity in the computational domain at all times, a new one is coming out of the orifice while an old one is leaving the domain. The distance traveled by the vortices is less as compared to that in Case 1, therefore the ensuing vortices move closer to each other. At the maximum expulsion stage, a new vortex is forming near the throat and the preceding vortex is $6.3d_o$ away from the throat exit. It is $7.6d_o$ in Case 1. Similar observation can be
made at maximum ingestion stage. The vortex travels $1.8d_o$ distance from the throat exit in Case 2 while this distance is $2.2d_o$ in Case 1. Furthermore, stronger vortices are observed for rectangular-shaped cavity.

Shown in Figure 4.13 are the velocity profiles in $x$- and $y$-direction for triangular-shaped cavity. Velocities are normalized with the local speed of sound.

![Velocity Profiles](image)

**Figure 4.13.** Case 2: Velocity profiles at the throat exit at four stages of one membrane oscillation cycle; a) $u$-velocities, b) $v$-velocities.

Figure 4.14 compares the streamwise velocity profiles at the throat exit for the rectangular-shaped and the triangular-shaped cavities at four stages of one membrane oscillation cycle.

The velocities exhibit the same profile characteristic for the ingestion phase. However, they display different characteristics near the wall region at the maximum expulsion stage; the velocity profile is narrower for the rectangular-shaped cavity. Also, a higher value of the maximum velocity is observed for Case 1 as compared to Case 2.
The mean jet velocities in the streamwise direction, $\bar{V}_{\text{jet}}$, are calculated to be 12.93 m/s and 12.56 m/s at the maximum expulsion stage for the rectangular- and triangular-shaped cavity, respectively.

![Comparison of velocity profiles at the four stages of membrane oscillation cycle](image)

Figure 4.14. Comparison of velocity profiles at the four stages of membrane oscillation cycle: a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.

4.1.3.2. Throat Height

In this subsection, we examine the effect of the parameter $h_o/d_o$ on the velocity profile and on the vortex dynamics. The throat width is kept at a fixed value of 500 μm which 

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
corresponds to unity in the computational domain. Figure 4.15 illustrates the throat geometries and computational grids with the close-ups to the throat region. In Case 3, the synthetic jet geometry does not have a throat so that the flow inside the cavity directly interacts with the external region (Fig. 4.15a). For Cases 4-8, the throat ratio is varied from a value of $h_o/d_o=0.5$ to $h_o/d_o=3$ (Fig. 4.15b).

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{throat_geometries.png}
\caption{Throat geometries: a) Case 3; b) Cases 4-8.}
\end{figure}

In Figure 4.16, the flow is illustrated via vorticity contours at the maximum expulsion stage. As seen in the figure, the ensuing vortex structure is different in Case 3 ($h_o/d_o=0$) when compared to the rest of the cases. The vortices are less strong and observed to move closer in Case 3. The distances of the first vortex from the throat exit are $3d_o$, $6.5d_o$, $6.2d_o$, $6.1d_o$, $5.9d_o$, and $5.8d_o$ for Cases 3, 4, 5, 6, 7, and 8, respectively. As the throat height is increased the boundary layer formed within the throat develops resulting in a smaller distance traveled by the vortices.
Comparisons of the $u$- and $v$-velocities at the throat exit between Cases 2-8 are given in Figures 4.17 and 4.18. The spanwise velocities change only slightly with increasing $h_o/d_o$ for Cases 2, 4-8 (Fig. 4.17). It is observed that the streamwise velocity profile has a different characteristic for $h_o/d_o=0$ (Case 3) at all four stages. For the cases with a throat, the streamwise velocity profiles are more like parabolic for lesser $h_o/d_o$ ratios at the maximum expulsion stage and the parabolic profile flattens for higher $h_o/d_o$ ratios. This
is consistent with the physical reasoning that the jet flow resembles the entrance flow in a channel so that the velocity exhibits a wider profile. Furthermore, there is a

![Graphs showing spanwise velocity profiles for Cases 2-8: a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.](image)

**Figure 4.17.** Spanwise velocity profiles for Cases 2-8; a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.

slight tendency to increase the spanwise velocity magnitude near the wall and also, the spanwise velocity magnitude decreases monotonically towards the centerline of the throat as we increase the throat height. Comparisons of the $v$ velocity at the rest of the stages of the cycle show that the overall shape of the profile as well as the velocity magnitude remains unchanged.
The mean jet velocities at the orifice exit, $V_{jet}$, at the maximum expulsion stage are presented in Table 4.2. As seen in Table 4.2, the calculated mean jet velocity has the maximum value for $h_o/d_o=0$. As for the Cases 2, 4-8, the flow through the throat with larger $h_o/d_o$ ratios is more developed and therefore the mean jet velocity at the throat exit increases. The relationship between the maximum jet velocity and the throat height is plotted in Figure 4.19.
Table 4.2. Mean jet velocities at the maximum expulsion stage for Cases 2-8.

<table>
<thead>
<tr>
<th>Case</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\bar{V}_{\text{jet}}$, m/s</td>
<td>12.562</td>
<td>12.987</td>
<td>12.563</td>
<td>12.681</td>
<td>12.739</td>
<td>12.755</td>
<td>12.788</td>
</tr>
</tbody>
</table>

Figure 4.19. Mean streamwise velocity at the orifice exit vs. orifice height.

Figure 4.20. Case 8: Velocity profiles at five locations: at maximum expulsion stage.

In Figure 4.20, the streamwise velocity profiles at five different locations along the throat are plotted at the maximum expulsion stage of the membrane oscillation cycle. It
is observed that the streamwise velocity is horned at the edges and as we go up towards the throat exit the profile gains a more parabolic characteristic.

4.1.3.3. Throat Width

In this subsection, the synthetic jet effectiveness is investigated for various values of throat widths. Throat width has been found to have an immense effect for improving the synthetic jet efficiency. We will investigate the effects of throat width again on the vortex dynamics and the mean jet velocity at the throat exit. Orifice width is varied from $0.2d_{ref}$ to $5d_{ref}$ while all other parameters are kept constant at their values of the baseline case. A value of $d_{ref}=5\times10^{-4}$ μm corresponds to 1 in the computational domain.

In Figure 4.21, the flow domains for the computational cases are illustrated via vorticity contours at the maximum expulsion stage for Cases 2, 9-11. For $d_o/d_{ref}=0.2$, an apparent vortex is formed and ejected from the throat exit creating a sequence of vortices, yet the vortex strength is observed to be relatively weak. Increasing the throat width ratio to $d_o/d_{ref}=1$ yields stronger and thus, more apparent vortex. Also, the distance traveled by the vortex is increased when compared to Case 9. Further increase in the $d_o/d_{ref}$ produces weaker vortices and lessens the vortex’s self-induced velocity so that the distance traveled by the vortex is reduced $d_o/d_{ref}=2$. As for Case $d_o/d_{ref}=5$, although there is a small vortex formed, it does not engender vortex shedding. It may be deduced here that the higher values as well as very small values of throat width will yield neither vortex formation nor vortex shedding.
Figure 4.21. Plot of vorticity contours for different $d_r/d_{ref}$ ratios: a) Case 9: 0.2, b) Case 2: 1.0, c) Case 10: 2.0, d) Case 11: 5.0

The mean jet velocities at the orifice exit, $V_{jet}$, at the maximum expulsion stage are presented in Table 4.3. Furthermore, the plot of the mean streamwise jet velocity at the throat exit calculated at the maximum expulsion stage for Cases 2, 9-11 vs. orifice width is shown in Figure 4.22. This plot further confirms the remarks concluded from the vorticity contour plots. The velocity magnitude first increases with increasing the orifice width; then a further increase results in a decrease in the velocity. Thus it is stated that the orifice width is an important parameter of the synthetic jet.

Table 4.3. Mean jet velocities at the maximum expulsion stage for Cases 2, 9-11.

<table>
<thead>
<tr>
<th>Case</th>
<th>2</th>
<th>9</th>
<th>10</th>
<th>11</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_{jet}$, m/s</td>
<td>12.56</td>
<td>9.22</td>
<td>6.16</td>
<td>0.71</td>
</tr>
</tbody>
</table>
4.1.3.4. Membrane Oscillation Amplitude

Another important design variable may be the membrane oscillation amplitude. In order to examine the effect of amplitude all other parameters are kept constant at their baseline values. A throat width of 500 μm corresponds to unity in the computational domain.

In Figure 4.23, the flow domains for Cases 1, 12-16 are illustrated via vorticity contours at the minimum volume stage of the membrane oscillation cycle. It is observed clearly from the plots that the vortex strength increases with less dissipation rates and vortex shedding becomes more visible as we increase the membrane oscillation amplitude. Furthermore, the distance traveled by the vortices emanating from the orifice increases with higher amplitude values. For smaller values of the amplitude, no apparent vortex formation is observed (Figs. 4.23a and 4.23b). In Case 14 (A/d₀=0.2) the self-induced velocity of the vortex is not sufficient to shed from the throat exit.
In Figure 4.24, a comparison between Cases 1 and 12-16 is given by the streamwise velocity plots at the throat exit for both expulsion and ingestion phases. Comparisons of the $v$-velocity at all the stages of the cycle show that the overall shape of the profile exhibits similar attributes for all computational cases. It is observed that the velocity magnitude increases with increasing the amplitude. However, the horned profile becomes clearer and the edges of the horn diverge from the centerline of the throat for the expulsion and the ingestion phases. This can be observed mainly at the maximum ingestion stage. Also interesting to note is that the velocity profile of Case 16 ($A/d_0=0.8$)
at the maximum expulsion stage differs from the rest of the profiles: negative velocities are realized in the proximity of the throat wall.

Figure 4.24. Cases 1, 12-16: Velocity profiles at the throat exit; a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.

The mean jet velocities at the orifice exit, $V_{jet}$, at the maximum expulsion stage are presented in Table 4.4. The relationship between the oscillation amplitude and the jet velocity is rather linear confirming the relationship given by Eq. 4.7 (Fig. 4.25).
Table 4.4. Mean jet velocities at the maximum expulsion stage for Cases 1, 12-16.

<table>
<thead>
<tr>
<th>Case</th>
<th>1</th>
<th>12</th>
<th>13</th>
<th>14</th>
<th>15</th>
<th>16</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\bar{V}_{jet}$, m/s</td>
<td>12.987</td>
<td>1.603</td>
<td>3.216</td>
<td>6.453</td>
<td>18.65</td>
<td>22.89</td>
</tr>
</tbody>
</table>

Figure 4.25. Mean streamwise velocity at the orifice exit vs. membrane oscillation amplitude.

4.1.3.5. Membrane Oscillation Frequency

Frequency dependency of the synthetic jet efficiency is examined by keeping all other parameters constant at their baseline case values while varying the membrane oscillation frequency. The frequency values are varied from 250 to 2000 Hz.

Figure 4.26 shows the vorticity contour plots of Cases 2, 17-23 at the maximum expulsion stage. For $f = 250$ Hz, although a very weak vortex is formed within the throat, it is not strong enough to break away from the throat exit. Further increasing the frequency creates stronger and thus, more visible vortices. Also, the distance traveled by the vortex increases with the higher values of frequencies.
Figure 4.26. Plot of vorticity contours for different frequencies: a) $f = 250$ Hz; b) $f = 500$ Hz; c) $f = 750$ Hz; d) $f = 1000$ Hz; e) $f = 1250$ Hz; f) $f = 1500$ Hz; g) $f = 2000$ Hz.

In Figure 4.27, a comparison between for Cases 2, 17-23 is given by the streamwise velocity plots at the throat exit at each stages of the cycle. Comparisons show that the overall shape of the profile exhibits similar attributes for all the cases. However, the velocity magnitude increases with increasing the membrane oscillation frequency.

The mean jet velocities at the orifice exit, $V_{jet}$, at the maximum expulsion stage are presented in Table 4.5. As it is observed in Figure 4.28, the relationship between the

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
oscillation frequency and the jet velocity is almost linear confirming the relationship given by Eq. 4.7.

Figure 4.27. Velocity profiles for Cases 2, 17-23: a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.

Table 4.5. Mean jet velocities at the maximum expulsion stage for Cases 2, 17-23.

<table>
<thead>
<tr>
<th>Case:</th>
<th>2</th>
<th>17</th>
<th>18</th>
<th>19</th>
<th>20</th>
<th>21</th>
<th>22</th>
<th>23</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \bar{V}_{\text{j}} ), m/s</td>
<td>12.562</td>
<td>3.127</td>
<td>6.268</td>
<td>9.421</td>
<td>15.86</td>
<td>19.03</td>
<td>22.60</td>
<td>26.37</td>
</tr>
</tbody>
</table>
4.1.4. Micron Level Synthetic Jets

Many studies have attempted to define a criterion to characterize the jet formation (Smith and Glezer [9]; Utturkar et al [27]; Holman et al [28]). However, in this subsection, we primarily focus on describing the effect of wall slip, resulting from the relatively larger Knudsen number flows associated with micro-sized geometries, on the exit jet velocity. Computations are carried out for the throat width values of $d_o=500 \, \mu\text{m}$, $d_o=50 \, \mu\text{m}$, and $d_o=5 \, \mu\text{m}$. The effect of the length scale on the velocity profiles and the vortex dynamics is discussed.

Shown in Table 4.6 are the definitions and results of the computational cases. A triangular-shaped cavity is selected for the actuator geometry and the geometric parameters are taken at their baseline values. Case 1 is calculated according to the continuum flow regime. For the rest of the computations, slip velocity boundary condition is implemented on the walls according to the slip flow regime.

![Figure 4.28. Mean streamwise velocity at the orifice exit vs. membrane oscillation frequency.](image)
Only the characteristic length scale is varied for Cases 1, 2, and 5 while all the other parameters of the synthetic jet are kept the same. In Case 3, the frequency is increased from 1 kHz to 10 kHz while all the other parameters are kept the same. In Case 4, the oscillation amplitude is doubled. The order of frequency is gradually increased for Case 6, and 7 whose jet velocities correspond to Case 2 and 3, respectively.

**Table 4.6. Computational cases: Definitions and results.** $d_0/h_0=1.$, $W/d_0=20.$, $H/d_0=4.$

\[
\text{Re} = \frac{\overline{V} \_\text{jet} \ d_0}{\nu}, \quad \text{St} = \sqrt{\frac{\omega \ d_0^2}{\nu}}, \quad \text{Str} = \frac{\omega \ d_0}{\overline{V} \_\text{jet} \ \text{Re}}, \quad k_r = \frac{fd_0}{a_\infty}
\]

<table>
<thead>
<tr>
<th>Case</th>
<th>$d_0$ (µm)</th>
<th>$f$ (kHz)</th>
<th>$A/d_0$</th>
<th>$Kn$</th>
<th>$\overline{V} _\text{jet}$</th>
<th>Re</th>
<th>St</th>
<th>Str</th>
<th>$k_r$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>500.</td>
<td>1.</td>
<td>0.4</td>
<td>0.0001</td>
<td>12.51</td>
<td>426.</td>
<td>~10</td>
<td>2.5</td>
<td>1.4 $\times 10^{-3}$</td>
</tr>
<tr>
<td>2</td>
<td>50.</td>
<td>1.</td>
<td>0.4</td>
<td>0.001</td>
<td>1.251</td>
<td>5.55</td>
<td>~1</td>
<td>2.5</td>
<td>1.4 $\times 10^{-4}$</td>
</tr>
<tr>
<td>3</td>
<td>50.</td>
<td>10.</td>
<td>0.4</td>
<td>0.001</td>
<td>12.65</td>
<td>43.1</td>
<td>~3</td>
<td>2.5</td>
<td>1.4 $\times 10^{-3}$</td>
</tr>
<tr>
<td>4</td>
<td>50.</td>
<td>10.</td>
<td>0.8</td>
<td>0.001</td>
<td>25.2</td>
<td>85.2</td>
<td>~3</td>
<td>1.25</td>
<td>1.4 $\times 10^{-3}$</td>
</tr>
<tr>
<td>5</td>
<td>5.</td>
<td>1.</td>
<td>0.4</td>
<td>0.01</td>
<td>0.127</td>
<td>0.04</td>
<td>0.1</td>
<td>2.5</td>
<td>1.4 $\times 10^{-5}$</td>
</tr>
<tr>
<td>6</td>
<td>5.</td>
<td>10.</td>
<td>0.4</td>
<td>0.01</td>
<td>1.276</td>
<td>0.44</td>
<td>~0.3</td>
<td>2.5</td>
<td>1.4 $\times 10^{-4}$</td>
</tr>
<tr>
<td>7</td>
<td>5.</td>
<td>100.</td>
<td>0.4</td>
<td>0.01</td>
<td>12.94</td>
<td>4.4</td>
<td>~1</td>
<td>2.5</td>
<td>1.4 $\times 10^{-3}$</td>
</tr>
</tbody>
</table>

Figure 4.29 shows the vorticity contour plots for the computations at the maximum expulsion stage. It is observed that vortex formation is highly sensitive to the scale of the characteristic length (Figs. 4.29 a, b, e). There is no vortex formation observed for Cases 2 and 5, whereas a sequence of vortices is shedding from the throat exit for Case 1. Thus, it is interpreted that decreasing the characteristic length scale while keeping all the other parameters fixed reduces the jet velocity (Table 4.6) followed by a diminution of the vortex formation.
As seen from the table, the vortex formation is observed for values of $V_{jet} > 12$, $Re > 43$, $St > 3$ (Cases 1, 3, 4, 7) and no vortex formation is observed for values of $V_{jet} < 12$, $Re < 5.5$, $St < 3$ (Cases 2, 5-6), although the $Str$ number is calculated to be the same for all cases except Case 4. Also, it is observed that the highest $Re$ number together with the highest $St$ number forms the strongest vortex and creates vortex shedding.

The spanwise velocity profiles are plotted for Cases 1, 3, and 7 in Figure 4.30. The membrane oscillation frequency is increased for Cases 3 and 7 as we reduce the order of the characteristic length scale in order to attain the same mean jet velocity. The effect of
the characteristic length scale can be clearly observed as a decrease in the spanwise velocity especially at the maximum ingestion stage.

![Graphs showing velocity profiles for different cases](image)

**Figure 4.30.** $u$-velocity profiles for Cases 1, 3, 7: a) maximum expulsion; b) minimum volume c) maximum ingestion; d) maximum volume.

The streamwise velocity profiles are plotted for Cases 1, 3, and 7 in Figure 4.31. The figure further confirms the velocities calculated at the maximum expulsion stage in Table 4.6. Yet, a rather distinguishing observation is the difference in the velocity profile characteristics especially at the maximum ingestion stage. The velocity plots reflect the effect of the $St$ number on the jet velocity profile such that the jet can choke the unsteady...
boundary layer which is developed in the throat. Thus, it is hypothesized that the smaller $St$ numbers can lead to no jet formation regardless of the value of the jet velocity.

\[
0.02 \quad V_J \quad 0
\]

\[
0.04 \quad 0.02 \quad 0.02 \quad 0.04 \quad -0.06 \quad -0.04 \quad -0.02 \quad 0
\]

**Figure 4.31.** $v$-velocity profiles for Cases 1, 3, 7; a) maximum expulsion; b) minimum volume; c) maximum ingestion; d) maximum volume.

### 4.1.5. Effect of Ambient Fluid Temperature on Synthetic Jet Flow

The effect of the ambient temperature on the vortex dynamics as well as the jet velocity profile is examined at two characteristic scales, $d_o=500 \, \mu m$ and $d_o=50 \, \mu m$. All other parameters are kept constant at their baseline values.
Definitions and the results of the computational cases are shown in Table 4.7. A rectangular shaped cavity is implemented for the synthetic jet geometry. In Cases 1-3, ambient temperatures are taken as $T= 15^\circ C$, $T= 40^\circ C$, and $T= 80^\circ C$, respectively for the characteristic length of $d_o=500 \, \mu m$. The corresponding cases are Cases 4-6 for $d_o=50 \, \mu m$.

<table>
<thead>
<tr>
<th>Case</th>
<th>$d_o$ (µm)</th>
<th>$f$ (kHz)</th>
<th>$T$ (°C)</th>
<th>$\bar{V}_{jet}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>500</td>
<td>1.</td>
<td>15</td>
<td>12.93</td>
</tr>
<tr>
<td>2</td>
<td>500</td>
<td>1.</td>
<td>40</td>
<td>13.19</td>
</tr>
<tr>
<td>3</td>
<td>500</td>
<td>1.</td>
<td>80</td>
<td>13.87</td>
</tr>
<tr>
<td>4</td>
<td>50</td>
<td>10.</td>
<td>15</td>
<td>12.74</td>
</tr>
<tr>
<td>5</td>
<td>50</td>
<td>10.</td>
<td>40</td>
<td>12.74</td>
</tr>
<tr>
<td>6</td>
<td>50</td>
<td>10.</td>
<td>80</td>
<td>12.74</td>
</tr>
</tbody>
</table>

As seen from the table, the velocities at the maximum expulsion stage is influenced by the ambient temperature for $d_o=500 \, \mu m$ whereas the velocities do not change with the temperature for $d_o=50 \, \mu m$.

The vorticity contour plots are shown in Figure 4.32. The distances traveled by the vortices are observed to lessen with high dissipation rates as we increase the temperature (Figs. 4.32 a, b, c and d, e, f). This phenomenon is more distinguished for cases with $d_o=500 \, \mu m$ when compared to the cases with $d_o=50 \, \mu m$. 

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The spanwise and the velocity profile plots extracted from the throat exit for Cases 1-3 and Cases 4-6 are shown at the maximum expulsion stage of the membrane cycle are shown in Figures 4.33 and 4.34, respectively. The effect of changing the ambient temperature is only observed for the streamwise velocities. The spanwise velocities remain unchanged.
Figure 4.33. $u$- and $v$- velocity profiles for Cases 1-3 at maximum expulsion stage.

Figure 4.34. $u$- and $v$- velocity profiles for Cases 4-6 at maximum expulsion stage.
4.2. Control of Separated Flow past a Backward-Facing Step in a Microchannel

In the flow past a backward facing step, as the sudden change in flow conditions results in a significant variation of the mean free path of the gas molecules and the wall shear stresses, the separation occurs at the base (Karniadakis and Beskok [2]) as shown in Figure 4.40. Subsequently, the change in the slip velocity changes the mass flow through the channel.

In the second part of the study, the synthetic jet actuator is investigated for its effectiveness as a flow control device for controlling the separation region in a two-dimensional microchannel with a backward facing step. For this purpose, a synthetic jet actuator with a rectangular-shaped cavity is placed downstream of the step where the separation occurs (Fig. 4.35). The synthetic jet is characterized by a number of actuation and geometric parameters which were mentioned in the previous section. The channel is characterized by its geometrical dimensions; inlet channel length, $L_{in}$, outlet channel length and height, $L_{out}$ and $h_c$, respectively, and the step height, $s$. The outlet channel height, $h_c$, also defines the characteristic length. The channel flow is characterized by an inlet velocity, $U_{in}$ and temperature, $T_{in}$. The fluid is characterized by its dynamic viscosity, $\mu$, and density, $\rho$.

A large number of test cases have been analyzed in order to find an improved design for the synthetic jet actuator. All the computations are carried out according to slip flow regime. Since an actuator is being designed herein to control flow separation, a quantitative indicator is needed for its mathematical formulation and the choice of this indicator is not trivial. Several metrics used by the authors in the past included energy.
Figure 4.35. Synthetic jet actuator in a microchannel with backward facing step.

loss, reattachment point and area of the separation region. Searching the literature, it was
found, for example, that Ravindran [85] and Desai and Ito [86] have successfully used
enstrophy as an objective function. Enstrophy is the square of vorticity integrated over a
region. Selection of enstrophy as the objective function is motivated by the fact that
potential flows (zero vorticity) are frictionless and incurs low energy dissipation.
Enstrophy constitutes a quantifiable measure of separation that involves recirculation. As
such, effectiveness of actuation in reducing separation can be quantified by analytically
monitoring the enstrophy of the separated region. Therefore, enstrophy within the
separated region is used as the objective function when optimizing the actuator:

$$ J_{enst} = \frac{1}{2T} \int_0^T \iint_{\Omega} \left| \nabla \times U \right|^2 d\Omega_s dt $$

(4.9)

where $T$ is the time period for one membrane oscillation cycle. The objective function is
calculated in $\Omega_s \subset \Omega$, where $\square$ is the area of the computational domain and $\square_s$ is the
region where the flow is separated, hence, recirculating. $\square_s$ is always computed at a
given instant. Then, an indicator of flow separation control effectiveness may be selected
as the percent reduction of enstrophy:
\[
\%\text{reduction} = \frac{(J_{ens})_{\text{reference}} - (J_{ens})}{(J_{ens})_{\text{reference}}} \times 100 \tag{4.10}
\]

It should be noted that the reference case does not have the actuator, hence, the value of \((J_{ens})_{\text{reference}}\) or \(D_s\) do not vary in time for the reference case. As for the controlled cases, \(D_s\) values are calculated at four instants during one membrane oscillation cycle after the flow field reaches a periodic behavior.

### 4.2.1. Computational Aspects

Figure 4.36a shows the computational domain without synthetic jet control (for clarity only every fourth grid line is shown). The computational domain consists of two blocks: upstream of the step (inlet channel) and downstream of the step (outlet channel) regions. The outlet channel height \(h_c\) is taken to be 12.5 \(\mu\)m and the step height \(s\) is taken as \(s/h_c=0.467\). The entry to the channel is simulated. The channel inlet is located at \(x/h_c=0.86\). The inlet and the outlet channel lengths are taken as 10\(h_c\) and 15\(h_c\), respectively. Nitrogen is chosen as the working fluid.

The inflow boundary condition is prescribed on the left side of the channel with \(M=0.135\), \(Re=33\) and inlet temperature of 330 °K. As an outflow boundary condition, an inlet-to-outlet pressure ratio of 1.37 is prescribed on the right side of the channel. The second order slip boundary conditions are employed at the walls. Channel walls are prescribed as adiabatic.

The numerical grid is created by using Gridgen V14 [87]. The inlet channel region has a grid size of 257*57 (horizontal*vertical) and the outlet channel region has a grid size of 673*113. Only one fourth of the domain is shown in Figure 4.36b. The grid is
non-uniform throughout the channel in horizontal and vertical directions in order to save grid points in the computational domain. The stretching is towards the step in both regions in order to capture the separation bubble more accurately.

Figure 4.36. Uncontrolled Case: a) Computational domain and boundary conditions; b) numerical grid.

Figure 4.37a demonstrates the computational domain of a controlled case with a single synthetic jet actuator placed downstream of the step where the separation occurs. A rectangular-shaped cavity synthetic jet actuator is placed downstream of the step in order to demonstrate control of flow separation in a microfluidic device using a micro synthetic jet.

The Cartesian grid used for the controlled case is shown in Figure 4.37b (for clarity only every fourth grid line is shown). The computational domain consists of there blocks: cavity (horizontal*vertical: 225*65), upstream of the step (257*57) and
downstream of the step (673*113) regions. The stretching within the cavity region is towards the throat of the synthetic jet actuator.

\[ \text{slip velocity b.c. with adiabatic wall} \]

**Figure 4.37.** A controlled Case: a) Computational domain and boundary conditions, b) numerical grid.

### 4.2.2. Computational Cases and Discussion

Initially, a very slow \( (M = 0.00001) \) flow in the channel is considered to ensure that the synthetic jet formation and vortex shedding are both realized (Edis *et al* [23]). The synthetic jet actuator is placed 10.35\( h_c \) from corner of the step and the orifice width is taken as 1.25 \( \mu \text{m} \) for this case. Slip boundary conditions are implemented since the jet Knudsen number, based on the jet velocity and the orifice width is \( Kn_{jet} \cong 0.06 \). Plot of
vorticity contours at four stages of the cycle are shown in Figure 4.38. An apparent vortex pair is observed to form and shed from the throat exit.

![Figure 4.38](image)

**Figure 4.38.** Plot of vorticity contours at four stages of one membrane cycle, $M=0.00001$; a) maximum expulsion; b) minimum volume, c) maximum ingestion, d) maximum volume.

![Figure 4.39](image)

**Figure 4.39.** The velocity profile at the orifice exit at four different stages of the cycle; a) $u$-velocity, b) $v$-velocity.

The velocity profiles at the orifice exit for the four membrane stages are plotted in Figure 4.39. Here the velocities are in nondimensional form. Reference velocity that is...
used for normalizing is the reference speed of the sound. Velocity profiles exhibit different characteristics for the expulsion and ingestion phases when compared to quiescent external flow cases. The velocity is jet-like at the maximum expulsion and minimum volume stages, whereas it is suction at the maximum ingestion and maximum volume stages.

The definitions of the test cases are shown in Table 4.8.

**Table 4.8.** Description of computational cases: \( h_c = 12.5 \mu m. \)

<table>
<thead>
<tr>
<th>Case no</th>
<th>( h_o/h_c )</th>
<th>( d_o/h_c )</th>
<th>( W/h_c )</th>
<th>( H/h_c )</th>
<th>( A/h_c )</th>
<th>( f ) (kHz)</th>
<th>( x_{jet}/h_c )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
<td>n/a</td>
</tr>
<tr>
<td>1</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>2</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.15</td>
</tr>
<tr>
<td>3</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.25</td>
</tr>
<tr>
<td>4</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.35</td>
</tr>
<tr>
<td>5</td>
<td>0.05</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>6</td>
<td>0.2</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>7</td>
<td>0.1</td>
<td>0.05</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>8</td>
<td>0.1</td>
<td>0.2</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>9</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>10</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>11</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>12</td>
<td>0.1</td>
<td>0.1</td>
<td>1.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>13</td>
<td>0.1</td>
<td>0.1</td>
<td>4.</td>
<td>0.4</td>
<td>0.08</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>14</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.02</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>15</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.04</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>16</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.16</td>
<td>500.</td>
<td>10.05</td>
</tr>
<tr>
<td>17</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>1000.</td>
<td>10.05</td>
</tr>
<tr>
<td>18</td>
<td>0.1</td>
<td>0.1</td>
<td>2.</td>
<td>0.4</td>
<td>0.08</td>
<td>1000.</td>
<td>10.05</td>
</tr>
<tr>
<td>19</td>
<td>0.1</td>
<td>0.1</td>
<td>4.</td>
<td>0.8</td>
<td>0.08</td>
<td>1000.</td>
<td>10.05</td>
</tr>
<tr>
<td>20</td>
<td>0.1</td>
<td>0.1</td>
<td>4.</td>
<td>0.8</td>
<td>0.16</td>
<td>1000.</td>
<td>10.05</td>
</tr>
</tbody>
</table>
In Table 4.9., the results of each case are presented. As a reference case, the flow without any actuator control is also computed and its area and enstrophy calculated in the separation region to be $\Omega_s = 0.128$ and $J_{ens} = 216.2$, respectively. Presented in Figure 4.40 is the stream traces for the solution of the reference case in slip flow regime.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>ref</td>
<td>0.128</td>
<td>216.2</td>
<td>0.128</td>
<td>216.2</td>
<td>0.128</td>
<td>216.2</td>
</tr>
<tr>
<td>1</td>
<td>0.087</td>
<td>272.88</td>
<td>0.01</td>
<td>0.038</td>
<td>0.106</td>
<td>122.53</td>
</tr>
<tr>
<td>2</td>
<td>0.096</td>
<td>276.88</td>
<td>0.003</td>
<td>0.228</td>
<td>0.114</td>
<td>161.68</td>
</tr>
<tr>
<td>3</td>
<td>0.100</td>
<td>295.31</td>
<td>0.</td>
<td>0.</td>
<td>0.105</td>
<td>134.83</td>
</tr>
<tr>
<td>4</td>
<td>0.108</td>
<td>322.99</td>
<td>0.</td>
<td>0.</td>
<td>0.103</td>
<td>111.89</td>
</tr>
<tr>
<td>5</td>
<td>0.100</td>
<td>310.60</td>
<td>0.</td>
<td>0.</td>
<td>0.110</td>
<td>133.41</td>
</tr>
<tr>
<td>6</td>
<td>0.081</td>
<td>243.95</td>
<td>0.013</td>
<td>1.958</td>
<td>0.093</td>
<td>106.64</td>
</tr>
<tr>
<td>7</td>
<td>0.121</td>
<td>249.90</td>
<td>0.052</td>
<td>47.72</td>
<td>0.103</td>
<td>175.41</td>
</tr>
<tr>
<td>8</td>
<td>0.030</td>
<td>125.83</td>
<td>0.</td>
<td>0.</td>
<td>0.001</td>
<td>0.099</td>
</tr>
<tr>
<td>9</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
<td>0.</td>
</tr>
<tr>
<td>10</td>
<td>0.049</td>
<td>136.54</td>
<td>0.</td>
<td>0.</td>
<td>0.075</td>
<td>84.92</td>
</tr>
<tr>
<td>11</td>
<td>0.090</td>
<td>244.63</td>
<td>0.023</td>
<td>6.925</td>
<td>0.085</td>
<td>101.81</td>
</tr>
<tr>
<td>12</td>
<td>0.074</td>
<td>158.98</td>
<td>0.001</td>
<td>0.027</td>
<td>0.068</td>
<td>118.57</td>
</tr>
<tr>
<td>13</td>
<td>0.091</td>
<td>283.85</td>
<td>0.003</td>
<td>0.119</td>
<td>0.087</td>
<td>69.572</td>
</tr>
<tr>
<td>14</td>
<td>0.096</td>
<td>253.88</td>
<td>0.026</td>
<td>8.558</td>
<td>0.055</td>
<td>80.490</td>
</tr>
<tr>
<td>15</td>
<td>0.082</td>
<td>282.74</td>
<td>0.</td>
<td>0.</td>
<td>0.067</td>
<td>48.505</td>
</tr>
<tr>
<td>16</td>
<td>0.126</td>
<td>250.06</td>
<td>0.049</td>
<td>47.95</td>
<td>0.112</td>
<td>166.15</td>
</tr>
<tr>
<td>17</td>
<td>0.109</td>
<td>257.92</td>
<td>0.027</td>
<td>9.391</td>
<td>0.107</td>
<td>143.09</td>
</tr>
<tr>
<td>18</td>
<td>0.074</td>
<td>342.16</td>
<td>0.</td>
<td>0.</td>
<td>0.121</td>
<td>117.13</td>
</tr>
<tr>
<td>19</td>
<td>0.118</td>
<td>212.43</td>
<td>0.051</td>
<td>49.85</td>
<td>0.105</td>
<td>72.072</td>
</tr>
<tr>
<td>20</td>
<td>0.061</td>
<td>298.63</td>
<td>0.</td>
<td>0.</td>
<td>0.069</td>
<td>54.464</td>
</tr>
</tbody>
</table>

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 4.40. Reference case: Separated flow without control in a channel with backward facing step.

For the controlled cases, $\Omega_s$ is the area of the separation region and $J_{ens}$ is the enstrophy calculated for the four important instants during one cycle of the membrane oscillation after the flow field reaches a periodic behavior. $\overline{\Omega}_s$ and $\overline{J}_{ens}$ represent the values averaged over one membrane cycle. As observed from Table 5.9, with synthetic jet control, the separation region and its enstrophy are mostly reduced as compared to the reference case. For some cases, the separation region is totally eliminated during the minimum volume stage (this point will be revisited with Fig. 5.43). Note that the enstrophy values are calculated to be zero as $\Omega_s$ vanishes at the corresponding stages. Further, the separation region appears to be more sensitive to the changes in the design variables at minimum volume stage rather than the other stages. During maximum expulsion stages of all cases, although the area of the separation bubble is reduced, the value of the enstrophy increases and receives its highest value when compared to all other stages. Also, no vortex formation is observed for any of the controlled cases. It may be hypothesized that the momentum of the channel flow has significant effect on the performance of the actuator.
In Cases 1-4 a synthetic jet actuator with the orifice aspect ratio $d_o/h_o=1$ and cavity aspect ratio of $W/H=5$ is placed at *four selected distances* from the base step. It has been clearly observed that the area of separation region is altered as a function of the jet location. In Figure 4.41, the flow obtained in Cases 1-4 are depicted via its stream traces. Although separation bubble is totally eliminated at the Min. Vol. stage for Cases 3-4, the overall reduction is achieved by Case 1.

![Figure 4.41](image)

**Figure 4.41.** Flow past a micro backward facing step with flow control illustrated via stream traces. Close-up of separation region; a) Case1, b) Case2, c) Case3, d) Case4.
The velocity profiles at the orifice exit for Case 1 are shown in Figure 4.42 at four different stages of the membrane cycle. Maximum velocities are observed to occur when the membrane is at its lowest (Max. Vol.) and highest (Min. Vol.) levels in contrast with the observations made for the quiescent external flow cases. Velocities at the maximum expulsion and maximum ingestion stages further confirm the results of Table 5.9. Considerable slip velocities are observed on the throat walls at the minimum volume and maximum volume stages.

![Graphs showing velocity profiles for different cases](image)

**Figure 4.42.** Case 1-4: The velocity profile at the orifice exit at four different stages of the cycle; a) maximum expulsion, b) minimum volume, c) maximum ingestion, d) maximum volume.
Thus, in Cases 5-20, the location of the synthetic jet is fixed at \( x_{\text{jet}} = 10.05 \). However, the cavity dimensions and the membrane's oscillation parameters are varied in order to demonstrate their effects on the separated flow. In Cases 5-6 and in Cases 7-9, computations have been carried out for two different orifice height ratios \( \frac{h_o}{h_c} \) and two different orifice width ratios \( \frac{d_o}{h_c} \), respectively, while fixing the cavity geometry and the parameters of the oscillating membrane. Then, the cavity height and the cavity width are tested or their effectiveness of separation control in Cases 10-11 and 12-13, respectively. Cases 14-16 and 17-18 are to observe the effect of the membrane oscillation amplitude and the membrane oscillation frequency, respectively. Finally, Cases 19-20 are tested in order to evaluate the effect of enlarging the cavity on the separation bubble. All the parameters are found to have favorable effects to change the performance of the synthetic jet actuator in terms of reducing the area of the separation bubble as well as the enstrophy integrated over the separated region.

It is clearly discernible from Table 5.9 that \( d_o \) has a major effect on the synthetic jet performance. The best control is achieved for Case 8 \( (d_o = 0.2h_c) \). The separated flow near the step at four important stages of the membrane oscillation cycle is illustrated in Figure 4.43 for the achieved best control. The separation bubble is totally eliminated at the minimum volume stage. Also, \( u \) velocity and \( v \) velocity profiles at the throat exit are shown in Figure 4.44. Similar profile characteristics with Cases 1-4 are observed for the streamwise velocity. Furthermore, the spanwise velocities at the jet exit are increasing in the positive \( x \) direction.

Moreover, the velocity vectors of the reference (no-control) case are shown in Figure 4.45 along with those of the achieved best control for comparison purposes.
Figure 4.43. Case 8: \(d_o=0.2h_c\). Flow control illustrated via stream traces and vorticity contours. Close-up of separation region; a) maximum expulsion, b) minimum volume, c) maximum ingestion, d) maximum volume.

Figure 4.44. Case 8: \(d_o=0.2h_c\). The velocity profile at the orifice exit at four different stages of the cycle; a) \(u\)-velocity, b) \(v\)-velocity.
Figure 4.45. Velocity vectors: a) Reference case, b) Case 8 at maximum expulsion stage, c) Case 8 at minimum volume stage, d) Case 8 at maximum ingestion stage, e) Case 8 at maximum volume stage.
The second best case is when \( d_o = 0.5h_c \) and the worst case is found to be for \( d_o = 0.05h_c \). As such, it is concluded that moderate values of the throat width are required to achieve better results and that increasing \( d_o \) has more favorable effect than decreasing it (Fig. 4.46).

![Figure 4.46. Case 1, 7-9: Effect of throat width on separation control.](image)

A larger reduction in both \( \Omega_s \) and \( J_{ens} \) for the throat height of \( h_o = 0.2h_c \) is achieved when compared to \( h_o = 0.05h_c \), although the separation bubble is managed to be totally eliminated during the maximum volume stage for \( h_o = 0.05h_c \) (Table 5.9). Also, the area of the separation bubble for \( h_o = 0.2h_c \) is observed to remain the same as \( h_o = 0.1h_c \), although more reduction in enstrophy is managed. The effect of the throat height on the performance of the synthetic jet is plotted in Figure 4.47.

As far as the performance of the synthetic jet is considered, decreasing the cavity height from \( H = 0.4h_c \) to \( H = 0.2h_c \) is shown to have more favorable effect. When the cavity height is increased to \( H = 0.8h_c \), the percent reduction obtained for enstrophy is remains the same, and the percent reduction of the separation area is worsened when compared to
the case of $H=0.4h_c$. The plots for the effect of the cavity height on separation control versus $\bar{\Omega}_z$ and $\bar{J}_{ens}$ are shown in Figure 4.48.

**Figure 4.47.** Case 1, 5-6: Effect of throat height on separation control.

The effect of the cavity width on the synthetic jet performance is shown in Figure 4.49. The maximum performance is attained when the cavity width is at its minimum.
value, $W=1h_c$. Increasing the cavity width from $W=2h_c$ to $W=4h_c$ has an increasing yet less favorable effect on separation control.

**Figure 4.49.** Case 1, 12-13: Effect of cavity width on separation control.

Figure 4.50 shows the effect of the membrane oscillation amplitude. The synthetic jet performance is increased by increasing the oscillation amplitude. This effect is attenuated when the amplitude is varied from $0.08h_c$ to $0.16h_c$. Also, the membrane

**Figure 4.50.** Case 1, 14-16: Effect of membrane oscillation amplitude on separation control.
oscillation frequency is observed to improve the performance of the actuator (Fig. 4.51). The percent reduction of $\overline{Q}_s$ is steeper than that of $\overline{J}_{ens}$.

The percent reduction of $\overline{Q}_s$ is steeper than that of $\overline{J}_{ens}$.

![Graph showing the percent reduction of $\overline{Q}_s$ and $\overline{J}_{ens}$ versus frequency.]

Figure 4.51. Case 1, 17, 18: Effect of membrane oscillation frequency on separation control.

Finally, the synthetic jet cavity size is doubled in height and width (Case 19). Yet, this enlargement does not create better results when compared to Case 1. Furthermore, this enlargement followed by an increase in amplitude as well helps improve the performance of the actuator (Case 20).
4.3. Thermal Management of Microelectronic Devices Using Synthetic Jet Actuators

In this subsection, a synthetic jet is utilized as a cooling device for thermal management of a single microchip placed in a 2D channel. Different synthetic jet configurations are tested to investigate the effectiveness of a synthetic jet for its thermal management. The study is done in a trial-and-error fashion and it presents altered main channel flow results for various synthetic jet configurations. The effect is that synthetic jet enhances mixing by imparting momentum to the channel flow thus manipulating the temperature field in a positive manner. Computations are carried out according to slip flow regime.

One of the two major categories of chip package is the single-chip package (SCP). In SCP, each chip is in a self contained unit (ASM [51]). A simple geometry of an SCP is shown in Figure 4.52 (Yeh and Chu [50]). As seen from the figure, a constant heat flux is dissipated from the top surface of the package. The heat is being conducted through the thickness of the microchip package and transferred to another environment from the bottom surface. There is no heat transfer through the side walls of the microchip package.

A single-microchip package placed in a two dimensional channel is idealized as a rectangular block. A schematic of the simplified geometry of the single microchip and the channel are shown in Figure 4.53. The channel is characterized by its geometrical dimensions; inlet channel length, $L_{in}$ and outlet channel length $L_{out}$. The channel height in for both the inlet and outlet channels is the same, $h_c$. The microchip is characterized by its height, $s$, length, $L_{chip}$, and the heat flux dissipated from the microchip surface, $q^{\text{w}}$. The channel height, $h_c$, also defines the characteristic length. The
channel flow is characterized by an inlet velocity, $U_{in}$ and temperature, $T_{in}$. The fluid is characterized by its dynamic viscosity, $\mu$, and density, $\rho$.

![Diagram of single-chip package geometry](image)

**Figure 4.52.** Single-chip package geometry (Yeh and Chu [50]).

![Diagram of simplified microchip geometry](image)

**Figure 4.53.** Schematic of the simplified microchip geometry in two-dimensional space.

In the present study, the mean temperature and heat transfer coefficient over the microchip surface are calculated for all computations so as to demonstrate the effectiveness of synthetic jet cooling. The heat transfer coefficient is written in its non-dimensional form as following (White [78]; Rumsey et al [79]):

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
\( c_h = \frac{k(\partial T / \partial n)}{\rho_{ref} |U_{ref}| C_p (T_t - T_{ref})} \) \hspace{1cm} (4.11)

In the above equation, \( k \) is the thermal conductivity, \( C_p \) is the specific heat at constant pressure, and \( T_{ref} \) is the reference total temperature. A positive value of \( c_h \) indicates heat flux toward wall.

The percent reduction of the mean temperature over the microchip surface calculated over one membrane cycle may be defined as an indicator of synthetic jet cooling effectiveness as follows:

\[
\%T \text{ reduction} = \frac{(\bar{T}_{chip} - 273)_{ref} - (\bar{T}_{chip} - 273)}{(\bar{T}_{chip} - 273)_{ref}} \times 100
\] \hspace{1cm} (4.12)

Where, the subscript "reference" denotes the case without controlled synthetic jet cooling. Similarly, the percent increase of the non-dimensional mean heat transfer coefficient from the microchip surface over one membrane cycle can be calculated as:

\[
\% (\bar{c}_{h,chip}) \text{ increase} = \frac{(\bar{c}_{h,chip})_{ref} - \bar{c}_{h,chip}}{(\bar{c}_{h,chip})_{ref}} \times 100
\] \hspace{1cm} (4.12)

For the demonstration of thermal management, synthetic jets are placed on the top wall of the microchannel and the results are discussed in in the following sections.

4.3.1. Computational Aspects

Figure 4.54a shows the computational domain and boundary conditions without synthetic jet cooling. The computational domain consists of three blocks: first block is for the inlet channel, second block is for the microchip channel and third block is for the outlet channel. For the present demonstration cases, the outlet channel height, \( h_c \), is 500 \( \mu \text{m} \) and the microchip height is taken as \( s/h_c=0.5 \). The microchip channel length is taken to
be $5h_c$. The inlet and outlet channel lengths are $15h_c$ and $20h_c$, respectively, so as to eliminate the entrance effects and outflow effects and in order to ensure a fully developed flow in the channel. Nitrogen is chosen as the working fluid.

The inlet boundary condition is defined on the left side of the domain with an inlet velocity of 0.5 m/s and an inlet temperature of 313.15°K. The inflow and outflow pressures are atmospheric and an inlet to outlet pressure ratio of unity is prescribed for $Re=14.67$ on the right side of the domain. A constant heat flux is imposed over the top wall surface of the microchip. It is assumed that there is no heat transfer through the side walls of the microchip and constant temperature that equals the inlet temperature, is imposed over the top and bottom walls of the channel. Also, slip velocity boundary condition is prescribed on the walls of the channel and on the surfaces of the microchip.

![Figure 4.54. Computational domain and boundary conditions for no synthetic jet cooling. (every fourth grid line shown for clarity).](image)

The numerical grid created by using Gridgen V14 [87] is shown in Figure 4.54b (only one fourth of the domain shown for clarity). The inlet channel region has a grid size of
657*129 (horizontal*vertical), the microchip channel and the outlet channel regions have grid sizes of 369*65 and 673*129, respectively. The grid is non-uniform throughout the channel in horizontal and vertical directions in order to save on computational cost. The stretching is towards the microchip in the inlet and outlet channel regions.

Figure 4.55a demonstrates the computational domain with a triangular-shaped cavity of a controlled thermal management case with a single synthetic jet actuator placed on top wall of the microchip channel.

The Cartesian grid used for a synthetic jet cooling configuration is shown in Figure 4.55b (for clarity, every fourth grid line is shown). The computational domain consists of two more blocks in addition to no-cooling configuration: cavity block (horizontal*vertical: 417*81) and the throat block (33*33). The stretching within the cavity region is towards the throat of the synthetic jet actuator.

**Figure 4.55.** Computational domain and boundary conditions for a synthetic jet cooling configuration. (every fourth grid line shown for clarity).
4.3.2. Computational Cases and Discussion

In this section, computations of controlled thermal management cases are presented. First, a case without synthetic jet cooling is discussed (Case 1). Case 1 demonstrates the $Re=14.67$ flow in the channel without control. In order to obtain an accurate approximation of the model, it is computed with no-slip boundary condition according to continuum flow regime and computed again with slip boundary condition according to slip flow regime (Table 4.10). The flow fields are illustrated via stream traces and temperature fields are demonstrated via temperature contours (Fig. 4.56).

<table>
<thead>
<tr>
<th>Case</th>
<th>Flow regime</th>
<th>$q''$ [W/m$^2$]</th>
<th>$T_{chip}$ (°K)</th>
<th>$c_{h,chip}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1a</td>
<td>slip</td>
<td>12.</td>
<td>346.01</td>
<td>-0.2423</td>
</tr>
<tr>
<td>1b</td>
<td>continuum</td>
<td>12.</td>
<td>351.34</td>
<td>-0.2434</td>
</tr>
</tbody>
</table>

Figure 4.56. Case 1: Flow and temperature fields; top: no-slip boundary condition, bottom: slip boundary condition.
Figure 4.57. Case 1: Comparison of temperature profiles over the top surface of microchip for slip and no-slip boundary conditions.

Temperature profiles over the top surface of the microchip for Cases 1a and 1b are plotted (Fig. 4.57) in order to observe the effect of the no-slip boundary conditions (Baysal and Aslan [19]): no-slip velocity creates friction on the surface, thus frictional heating increases the maximum wall temperature, $T_{\text{chip}}^{\text{max}}$, from 363 K to 371 K for the heat flux value of $q'' = 12$ W/m$^2$ imposed over the microchip surface. Furthermore, the mean temperatures, $\overline{T}_{\text{chip}}$, calculated over the microchip surface in continuum and slip flow regimes are 351 K and 346 K, respectively. The non-dimensional heat transfer coefficient per unit area from the microchip surface is also calculated for Case 1a and it corresponds to a value of $\overline{\tau}_{h,\text{chip}} = -0.240978$. Note that a positive sign for the heat transfer coefficient indicates heat transfer towards the wall. Thus, the forthcoming cases with
synthetic jet cooling will be computed according to slip flow regime. Therefore, $\bar{T}_{\text{chip}} = 346$ and $\bar{e}_{h,\text{chip}} = -0.240978$ are selected as reference temperature and reference heat transfer coefficient per unit area, respectively, when calculating the percent reduction in $\bar{T}_{\text{chip}}$ and percent increase of $\bar{e}_{h,\text{chip}}$ for the corresponding controlled cases computed (Eqs. 4.11 and 4.12).

For all the controlled cases, the following geometric and actuation parameters of the synthetic jet are kept the same and they correspond to $d_o/h_c=0.1$, $H/h_c=0.4$, $W/h_c=2$, $A/h_c=0.08$ and $f =10$ kHz. The membrane oscillation frequency and the membrane oscillation amplitude are selected to be 10 kHz and $A/h_c=0.08$, respectively. They are found to be the minimum values together required in order to ensure that the synthetic jet formation with vortex shedding criterion (Edis et al [23]) is satisfied for the chosen micro size. Also, it has been found that placing the synthetic jet on the bottom wall has no significant influence on cooling of the microchip surface. Therefore, in all computational cases, the synthetic jets are positioned on the top wall of the microchannel.

Computational cases for single synthetic jet cooling are shown in Table 4.11. Also, a number of test cases are carried out in order to investigate the effectiveness of the synthetic jet in dissipating the heat from the microchip surface. Table 4.12 shows the test cases without synthetic jet cooling for various heat fluxes. The calculated mean temperature and mean heat coefficient values of these cases will be used as reference values for the corresponding controlled thermal management cases (Table 4.11, Cases 12-16).

In Table 4.13, the computational cases for multiple synthetic jet configurations are tabulated. Finally, the results of all the computational cases are shown in Table 4.14.
Table 4.11. Definitions of computational cases for single synthetic jet control, \( f = 10 \text{ kHz} \).

<table>
<thead>
<tr>
<th>Case</th>
<th>( x_{\text{jet}} )</th>
<th>( h_d/d_o )</th>
<th>( \dot{q}^* ) [W/m²]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>14.70</td>
<td>0.</td>
<td>12.</td>
</tr>
<tr>
<td>3</td>
<td>17.50</td>
<td>0.</td>
<td>12.</td>
</tr>
<tr>
<td>4</td>
<td>16.25</td>
<td>0.</td>
<td>12.</td>
</tr>
<tr>
<td>5</td>
<td>18.75</td>
<td>0.</td>
<td>12.</td>
</tr>
<tr>
<td>6</td>
<td>19.25</td>
<td>0.</td>
<td>12.</td>
</tr>
<tr>
<td>7</td>
<td>17.50</td>
<td>0.5</td>
<td>12.</td>
</tr>
<tr>
<td>8</td>
<td>17.50</td>
<td>1.0</td>
<td>12.</td>
</tr>
<tr>
<td>9</td>
<td>17.50</td>
<td>1.5</td>
<td>12.</td>
</tr>
<tr>
<td>10</td>
<td>17.50</td>
<td>1.0(directed)</td>
<td>12.</td>
</tr>
<tr>
<td>11</td>
<td>17.50</td>
<td>1.0(nozzlelike)</td>
<td>12.</td>
</tr>
<tr>
<td>12</td>
<td>17.50</td>
<td>1.0(nozzlelike)</td>
<td>4.5</td>
</tr>
<tr>
<td>13</td>
<td>17.50</td>
<td>1.0(nozzlelike)</td>
<td>15.</td>
</tr>
<tr>
<td>14</td>
<td>17.50</td>
<td>1.0(nozzlelike)</td>
<td>18.</td>
</tr>
<tr>
<td>15</td>
<td>17.50</td>
<td>1.0(nozzlelike)</td>
<td>24.</td>
</tr>
<tr>
<td>16</td>
<td>17.50</td>
<td>1.0(nozzlelike)</td>
<td>27.</td>
</tr>
</tbody>
</table>

Table 4.12. No-control cases for various heat fluxes.

<table>
<thead>
<tr>
<th>( \dot{q}^* ) [W/m²]</th>
<th>( T_{\text{chip}} ) [°K]</th>
<th>( \bar{c}_{h,\text{chip}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.5</td>
<td>325.84</td>
<td>-0.2405</td>
</tr>
<tr>
<td>15.</td>
<td>353.79</td>
<td>-0.2432</td>
</tr>
<tr>
<td>18.</td>
<td>361.39</td>
<td>-0.2441</td>
</tr>
<tr>
<td>24.</td>
<td>376.16</td>
<td>-0.2459</td>
</tr>
<tr>
<td>27.</td>
<td>383.34</td>
<td>-0.2468</td>
</tr>
</tbody>
</table>

Table 4.13. Definitions of computational cases for multiple synthetic jet control \( \dot{q}^* = 12 \text{ W/m²} \), \( h_d/d_o = 0 \).

<table>
<thead>
<tr>
<th>Case</th>
<th>JET LOCATION</th>
<th>( f ) [kHz]</th>
<th>PHASE ANGLE, ( \Phi )</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>JET 1 JET 2 JET 3</td>
<td>JET 1 JET 2 JET 3</td>
<td>JET 1 JET 2 JET 3</td>
</tr>
<tr>
<td>17</td>
<td>14.70 17.50 n/a</td>
<td>10. 10. n/a</td>
<td>0. 0. n/a</td>
</tr>
<tr>
<td>18</td>
<td>16.25 18.75 n/a</td>
<td>10. 10. n/a</td>
<td>0. 0. n/a</td>
</tr>
<tr>
<td>19</td>
<td>16.25 19.25 n/a</td>
<td>10. 10. n/a</td>
<td>0. 0. n/a</td>
</tr>
<tr>
<td>20</td>
<td>16.45 18.55 n/a</td>
<td>10. 10. n/a</td>
<td>0. 0. n/a</td>
</tr>
<tr>
<td>21</td>
<td>16.25 18.75 n/a</td>
<td>10. 10. n/a</td>
<td>0. 180. n/a</td>
</tr>
<tr>
<td>22</td>
<td>16.25 18.75 n/a</td>
<td>10. 6 n/a</td>
<td>0. 180. n/a</td>
</tr>
<tr>
<td>23</td>
<td>16.25 18.75 n/a</td>
<td>6. 10. n/a</td>
<td>0. 180. n/a</td>
</tr>
<tr>
<td>24</td>
<td>15.45 17.50 19.55</td>
<td>10. 10. 10.</td>
<td>0. 0. 0.</td>
</tr>
<tr>
<td>25</td>
<td>15.45 17.50 19.55</td>
<td>10. 10. 10.</td>
<td>0. 180. 0.</td>
</tr>
<tr>
<td>26</td>
<td>15.45 17.50 19.55</td>
<td>10. 10. 10.</td>
<td>0. 180. 0.</td>
</tr>
<tr>
<td>27</td>
<td>15.45 17.50 19.55</td>
<td>10. 10. 10.</td>
<td>0. 0. 180.</td>
</tr>
</tbody>
</table>

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Table 4.14. Results of computational cases of synthetic jet cooling.

<table>
<thead>
<tr>
<th>Case</th>
<th>$T_{\text{chip}}$ (°K)</th>
<th>$% \bar{T}$ reduction</th>
<th>$\bar{c}_{h,\text{chip}}$</th>
<th>$% \bar{c}_{h,\text{chip}}$ increase</th>
</tr>
</thead>
<tbody>
<tr>
<td>1a</td>
<td>346.01</td>
<td>n/a</td>
<td>-0.2410</td>
<td>n/a</td>
</tr>
<tr>
<td>2</td>
<td>346.53</td>
<td>-0.70</td>
<td>-0.2268</td>
<td>-0.95</td>
</tr>
<tr>
<td>3</td>
<td>334.01</td>
<td>16.48</td>
<td>-0.4713</td>
<td>95.6</td>
</tr>
<tr>
<td>4</td>
<td>334.86</td>
<td>15.31</td>
<td>-0.4178</td>
<td>73.4</td>
</tr>
<tr>
<td>5</td>
<td>335.43</td>
<td>14.53</td>
<td>-0.3951</td>
<td>63.9</td>
</tr>
<tr>
<td>6</td>
<td>336.88</td>
<td>12.54</td>
<td>-0.3604</td>
<td>49.6</td>
</tr>
<tr>
<td>7</td>
<td>334.08</td>
<td>16.35</td>
<td>-0.4805</td>
<td>99.4</td>
</tr>
<tr>
<td>8</td>
<td>333.48</td>
<td>17.17</td>
<td>-0.5020</td>
<td>108.3</td>
</tr>
<tr>
<td>9</td>
<td>332.91</td>
<td>17.96</td>
<td>-0.5095</td>
<td>111.4</td>
</tr>
<tr>
<td>10</td>
<td>334.06</td>
<td>16.38</td>
<td>-0.4866</td>
<td>101.9</td>
</tr>
<tr>
<td>11</td>
<td>332.90</td>
<td>17.98</td>
<td>-0.5129</td>
<td>112.8</td>
</tr>
<tr>
<td>12</td>
<td>321.00</td>
<td>9.19</td>
<td>-0.4616</td>
<td>91.9</td>
</tr>
<tr>
<td>13</td>
<td>337.50</td>
<td>20.14</td>
<td>-0.5203</td>
<td>113.9</td>
</tr>
<tr>
<td>14</td>
<td>342.10</td>
<td>21.85</td>
<td>-0.5259</td>
<td>115.4</td>
</tr>
<tr>
<td>15</td>
<td>351.20</td>
<td>24.23</td>
<td>-0.5330</td>
<td>116.7</td>
</tr>
<tr>
<td>16</td>
<td>355.60</td>
<td>25.13</td>
<td>-0.5361</td>
<td>117.2</td>
</tr>
<tr>
<td>17</td>
<td>334.76</td>
<td>15.41</td>
<td>-0.4203</td>
<td>74.4</td>
</tr>
<tr>
<td>18</td>
<td>328.84</td>
<td>23.56</td>
<td>-0.6232</td>
<td>158.6</td>
</tr>
<tr>
<td>19</td>
<td>330.32</td>
<td>21.52</td>
<td>-0.6018</td>
<td>149.7</td>
</tr>
<tr>
<td>20</td>
<td>328.96</td>
<td>23.38</td>
<td>-0.6118</td>
<td>153.8</td>
</tr>
<tr>
<td>21</td>
<td>327.76</td>
<td>25.04</td>
<td>-0.6769</td>
<td>180.9</td>
</tr>
<tr>
<td>22</td>
<td>333.76</td>
<td>17.33</td>
<td>-0.4024</td>
<td>66.99</td>
</tr>
<tr>
<td>23</td>
<td>330.53</td>
<td>21.23</td>
<td>-0.4848</td>
<td>101.2</td>
</tr>
<tr>
<td>24</td>
<td>330.36</td>
<td>21.46</td>
<td>-0.6239</td>
<td>158.9</td>
</tr>
<tr>
<td>25</td>
<td>329.80</td>
<td>22.23</td>
<td>-0.6639</td>
<td>175.5</td>
</tr>
<tr>
<td>26</td>
<td>327.68</td>
<td>25.15</td>
<td>-0.7121</td>
<td>195.5</td>
</tr>
<tr>
<td>27</td>
<td>328.47</td>
<td>24.06</td>
<td>-0.5734</td>
<td>137.9</td>
</tr>
</tbody>
</table>
4.3.2.1. Single Synthetic Jet Cooling

4.3.2.1.1. Effect of Jet Location

In Cases 2-6, five different stations for a single synthetic jet actuator are pre-selected in search for the best location in a trial-and-error manner (Table 4.11). Obtained results are shown in Table 4.14. As the test geometry, a triangular shaped cavity without a throat is selected for the synthetic jet actuator (Fig. 4.58).

![Figure 4.58. Selected synthetic jet actuator geometry to test effect of jet location and its numerical grid.](image)

In Case 2, synthetic jet is positioned upstream of the microchip and $0.30 h_c$ away from the microchip channel entrance. Then, four other different locations on the top wall of the microchip channel (Cases 3-6) are selected in order to test the effect of the jet location (Fig. 4.59). In Case 6, the synthetic jet is positioned where the maximum temperature occurs on the microchip surface. It is observed that the synthetic jet actuation creates a pair of vortices that is forming a circulation region in the channel (Fig. 4.60). This circulation region of Case 2 differs from that of Cases 3-6. It is found to create an
obstruction for the fluid flow in the channel resulting in a negative effect on thermal management when placed upstream of the microchip channel.

Figure 4.59. Five stations to test the effect of the location.

Figure 4.60. Instantaneous vorticity contour plots demonstrating the circulation region. Top: Case 2, bottom: Case 3.

In Figure 4.61, the instantaneous temperature fields are shown for each case with the maximum and minimum temperatures over the microchip surface. Also, mean
temperature profiles over the microchip surface are plotted in Figure 4.62 for a better comparison.

Figure 4.61. Cases 2-6: Instantaneous temperature fields.
The best control is achieved with a reduction of 16.48% in mean temperature over the microchip surface when synthetic jet is placed midway of the chip length (Case 3 in Table 4.11). Also, an increase of 95.6% is obtained for the heat transfer coefficient.

![Figure 4.62. Temperature plots over the microchip surface; Cases 1-6.](image)

**4.3.2.1.2. Effect of Throat Geometry**

Six different configurations of synthetic jet throat geometry are tested in Cases 3 and 7 through 11 (Fig. 4.63). Computations are carried out in order to evaluate the effect of the throat geometry on the thermal management of microchip cooling. The definitions and the results of the cases are tabulated in Tables 4.11 and 4.14, respectively. In all cases, the placement of the synthetic jet is fixed at its best location $x_{jet}=17.5$. 

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Case 3 has no throat, that is, \( \frac{h_o}{d_o} \) ratio is 0. This ratio is varied from 0.5 to 1.5 for Cases 7 through 9. As can be seen from Table 4.14, more effective cooling is achieved as \( \frac{h_o}{d_o} \) ratio is increased. A reduction of 17.96% in the mean temperature over the microchip surface is managed with \( \frac{h_o}{d_o} \) ratio of 1.5. In Case 10, the synthetic jet is directed towards the exit of the channel aiming more effective synthetic jet cooling. Yet, the percent reduction of temperature obtained for the directed jet is less when compared to the non-directed one (Case 8) (see Table 4.14). Then, a nozzle-like geometry is implemented with \( \frac{h_o}{d_o} \) ratio of 1.0 (Case 11) for the synthetic jet geometry. The highest heat transfer rate is achieved with a temperature decrease of 17.98 for this case. The close-up of temperature fields for each case is shown in Figure 4.63.

![Figure 4.63. Cases 3, 7-11: Different throat geometries with a close-up of temperature field.](image-url)
The membrane cycle-history of the nondimensional mean heat transfer coefficient for Case 11 is plotted in Figure 4.64. The negative value of the heat transfer coefficient per unit area indicates heat transfer outwards from the surface. While a steep increase in heat transfer coefficient is observed until the 10th cycle is reached, an almost steady state level is reached at the end of 20th membrane cycle.

![Figure 4.64. Convergence history of Case 11: Heat transfer coefficient per unit area versus membrane cycle number.](image)

4.3.2.1.3. Jet Cooling for Various Heat Fluxes

Different heat flux values, $q'' = k \frac{\partial T}{\partial n}$, are imposed over the microchip surface in order to investigate the effectiveness of synthetic jet cooling. A triangular-shaped cavity with a nozzle-like throat is used for the computations. First, computations without synthetic jet cooling are carried out according to slip flow regime: In Case 1, a value of
\( \hat{q}'' = 12 \text{ W/m}^2 \) was considered. In Cases 12-16, heat fluxes of 3 W/m\(^2\), 15 W/m\(^2\), 18 W/m\(^2\), 24 W/m\(^2\), and 27 W/m\(^2\) are imposed, respectively, over the top surface of the microchip to test thermal management. The mean temperatures as well as the nondimensional mean heat transfer coefficients over the microchip surface are tabulated in Table 4.12. The plots of \( \overline{T}_{\text{chip}} \) and \( \overline{c}_{h,\text{chip}} \) versus \( \hat{q}'' \) are shown in Figure 5.65. %T reduction and %\( \overline{c}_{h,\text{chip}} \) increase are calculated for controlled cases as shown in Table 4.14.

\[
\begin{array}{l}
\begin{array}{c}
\text{Figure 5.65. Left: Mean temperature over the microchip surface versus heat flux; right: heat transfer coefficient per unit area versus heat flux.}
\end{array}
\end{array}
\]

In Figure 4.66, the instantaneous temperature fields with the minimum and maximum temperatures over the microchip surface are shown for Cases 12-16. The captured instants correspond to the maximum expulsion stage.

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The percent increase in heat transfer coefficient versus heat flux is plotted in Figure 4.67 for Cases 11-16. As can be seen from the figure, the effect of synthetic jet for thermal management is strengthening as the heat from the surface increases.

Figure 4.66. Cases 12-16: Instantaneous temperature fields at maximum expulsion stage.
4.3.2.2. Multiple Synthetic Jet Cooling

4.3.2.2.1. Cooling with Two Jets

4.3.2.2.1.1. Effect of Jet Locations

Four different configurations are considered as shown in Figure 4.68. For each case, two synthetic jet actuators oscillating at the same phase angle are placed on the top wall of the 2D channel. The simplest geometry, that is triangular shaped cavity without a throat, is considered for the computations (Fig. 4.58). In the first setup (Case 17), the first jet is placed downstream of the microchip and the second one is placed at mid-length of the microchip. The orifice centers are $2.8h_c$ apart. For Cases 18, 19, and 20, both jets are
positioned on the top wall of the microchip channel with the distances between the orifice centers being $2.5h_c$, $3h_c$, $2.1h_c$, respectively.

Figure 4.68. Four configurations of synthetic jets; a) Case 17, b) Case 18, c) Case 19, d) Case 20.
In Figure 4.69, temperature fields are depicted via temperature contours. The instantaneous temperature profiles over the microchip surface are plotted in Figure 4.70.

Figure 4.69. Instantaneous temperature fields via temperature contours; a) Case 17, b) Case 18, c) Case 19, d) Case 20.
It is observed that increasing the number of synthetic jets improves the cooling. The best controlled cooling with a reduction of 23.56% in temperature and an increase in heat transfer coefficient of 158.6% is obtained in Case 18. As it can be seen from the plots as well as from Table 4.14 that Jet 1 of Case 17 has a negative effect of increasing the temperature over the surface. This also has been observed with single synthetic jet actuator placement in Case 2 (Table 4.14).

4.3.2.2.1.2. Effect of Phase Angle

In this subsection, the effect of phasing the membrane oscillations of the synthetic jets is considered (Table 4.13). The case of two synthetic jets oscillating with the same frequencies of 10 kHz (Case 18) is taken as the base case. First, two synthetic jets are
oscillated with a phase lag but keeping the membrane oscillation frequencies the same at \( f = 10 \text{ kHz} \) (Case 21). Then, membrane oscillation frequencies of \( f_{\text{Jet}1} = 10 \text{ kHz} \) and \( f_{\text{Jet}2} = 6 \text{ kHz} \) are applied in Case 22. These frequency values are switched for Case 23.

Shown in Fig. 4.71 are the temperature fields of Cases 21-23 illustrated via temperature contours. It can be clearly seen that phasing the membranes helps increase the heat transfer rate from the microchip surface. Another conclusion that may be drawn here is that the effect of the membrane oscillation frequency of Jet 2 is more consequential than that of Jet 1.

\[
\begin{align*}
\text{(a)} & \\
& (T_{\text{chip}}^{\text{min}}, T_{\text{chip}}^{\text{max}}) = (313.5, 355.06) \\
\text{(b)} & \\
& (T_{\text{chip}}^{\text{min}}, T_{\text{chip}}^{\text{max}}) = (313.5, 356.6) \\
\text{(c)} & \\
& (T_{\text{chip}}^{\text{min}}, T_{\text{chip}}^{\text{max}}) = (313.7, 358.6)
\end{align*}
\]

Figure 4.71. Cases 21-23: Instantaneous temperature fields illustrated via temperature contours.
The instantaneous temperature profiles of Cases 1, 18, 21-23 over the microchip surface at the maximum expulsion stage are plotted in Figure 4.72 for comparison purposes.

**Figure 4.72.** Temperature plots over the microchip surface; Cases 1, 18, 21-23.

**Figure 4.73.** Case 21: Instantaneous vorticity contours demonstrating the circulation region.
Shown in Figure 4.73 is the instantaneous flow field of Case 21 illustrated via vorticity contours and velocity vectors. Nondimensional mean heat transfer coefficient calculated over each cycle is plotted in Figure 4.74 for Case 21. Steady-state has been reached in 20 cycles.

\[ \bar{c}_{h,chip} \]

Figure 4.74. Case 21: Heat transfer coefficient per unit area versus membrane cycle number.

4.3.2.2.2. Cooling with Three Jets

In this subsection, results are introduced when three synthetic jets are placed in tandem for cooling (Cases 24-27). The jets are placed on the top wall of the microchip channel such that the distance between two consecutive jets are the same and equals to 2.05\( h_e \). Figure 4.75 shows the configuration with the placement of the jets.
Results obtained are shown in Table 4.14. It is interesting to note that adding one more jet to the configuration does not yield much better results in terms of mean temperature over the microchip surface as one would intuitively expect. On the contrary, the best cooling achieved with three synthetic jets (Case 26) is almost identical to those with one (Case 16) and two jets (Case 21). Yet, it is also interesting to observe that the best result is obtained with three jets in regard to the percent increase in mean heat transfer rate from the microchip surface.

Shown in Figure 4.76 are the temperature fields of Cases 24-27 illustrated via temperature contours. In Case 24, the jets are oscillating without a phase lag. In Cases 25, 26 and 27, Jet 1, Jet 2, and Jet 3, respectively, are oscillated at a 180° phase angle. It is observed that oscillating Jet 2 with a different phase angle results in better cooling (Case 26).

The flow fields are illustrated via vorticity contour in Figure 4.77. The difference in created circulation regions by each of the jets can be clearly seen from the plots.
Figure 4.76. Instantaneous temperature fields illustrated via temperature contours. a) Case 24, b) Case 25, c) Case 26, d) Case 27.
Figure 4.77. Instantaneous vorticity contours demonstrating the circulation region. a) Case 24, b) Case 25, c) Case 26, d) Case 27.
CHAPTER V
CONCLUSIONS

The utilization of synthetic jet actuators for manipulating fluid flow has been shown by previous researchers for macro- and mini-scale applications both computationally and experimentally. However, many issues remain unresolved for micron-scale synthetic jets and their applications to micron-scale problems. Also, although studies have shown improved results of synthetic jet applications, for instance, to flow separation and cooling of electronic devices, our understanding of what affects the performance of the synthetic jet is very much incomplete. This includes a high-fidelity modeling of the actuator which requires the modeling of the flow inside the cavity as well as the realistic modeling of the membrane. In addition, numerical approaches executed by other researchers lack the proper modeling of the flow which may require the implementation of particular boundary conditions at micron scales. Finally, the compressibility effects encountered in micron-scale devices has to be taken account.

The present study consists of three main parts: synthetic jet formation and evolution, synthetic jets to control flow separation, and synthetic jets for cooling of a microelectronic chip. A conventional compressible Navier-Stokes solver is used to model the flow domain. However, its boundary conditions are modified to account for the slip velocity and temperature jump conditions encountered in MEMS geometries for a Kn number range of 0.001 to 0.1. For validation purposes, simulations have been successfully performed for a micro channel, a micro filter and
a backward facing step. The velocity, temperature and pressure fields computed in the present study agree very well with the analytical formula and other independent computations available in the literature.

In the first part of the study, numerical simulations are designed to examine the effectiveness of a two-dimensional synthetic jet discharging into a quiescent medium. The membrane motion is modeled in a realistic manner as a moving boundary using a moving-deforming mesh simulation so as to accurately compute the flow inside and outside of the actuator cavity. The geometric and actuation parameters of the actuator (cavity geometry, $W$, $d_o$, $h_o$, $A$, $f$) as well as the micron-scale effects are studied in terms of how they affect the jet velocity and the vortex dynamics. The following significant conclusions are drawn:

- Maximum positive and maximum negative jet velocities at the throat exit are observed at the maximum expulsion stage and the maximum ingestion stages, respectively.

- Cavity shape is found to have considerable effect on the jet velocity characteristics at the throat exit as well as on the vortex dynamics. In triangular shaped cavity cases, the ensuing vortices are observed to move slower and thus are closer to each other as compared to the rectangular shaped cavity cases.

- The ensuing vortex structure is observed to be different for the case without a throat ($h_o/d_o=0$) as compared to the cases with a throat: The vortices are weaker and move slower for the case without the throat. Also, as the throat height is increased, the boundary layer within the throat
develops, so the distance traveled by the vortices decreases. The streamwise velocity profile also has a different characteristics for $h_0/d_o=0$ at all four stages of one membrane cycle. For the geometry with a throat, the streamwise velocity profiles are more like parabolic for lesser $h_0/d_o$ ratios and the parabolic profile flattens for higher $h_0/d_o$ ratios.

- Throat width, $d_o$, is also shown to be an important parameter for the synthetic jet effectiveness. It is observed that the relatively higher values as well as smaller values of throat width will yield neither vortex formation nor vortex shedding.

- Another important design variable is the membrane oscillation amplitude. The vortex strength increases accompanied with less dissipation rates and vortex shedding becomes more visible as we increase the amplitude. Also, a linear relationship is observed between the amplitude and the resulting mean jet velocity.

- Membrane oscillation frequency also has a similar effect on the jet velocity. The jet profile increases in magnitude as we increase the frequency. The relationship between the frequency and the jet velocity is also linear.

- The effect of the length scale on the velocity profiles and the vortex dynamics is considered as well. Decreasing the characteristic length scale reduces the jet velocity followed by a diminution of the vortex formation. It is observed that the vortex formation strongly depends on the jet Reynolds number and the Stokes number: The highest Re number together
with the highest Stokes number forms the strongest vortex and creates vortex shedding. The Strouhal number for a case with vortex formation is calculated to be the same as that for a case without vortex formation. Thus, it is concluded that Strouhal number alone is not sufficient to determine a criterion for the vortex formation in general.

- The effect of the ambient temperature is also examined at two different characteristic scales. The velocities at the maximum expulsion stage are influenced by the ambient temperature for bigger characteristic scales, whereas the velocities do not change with temperature for smaller length scales. In addition, distances traveled by the vortices are observed to lessen with high dissipation rates as we increase the temperature.

In the second part of the study, a micro synthetic jet is proposed as a flow control device. For demonstration, a flow past a micro backward-facing step is considered. A large number of test cases are analyzed. Slip boundary conditions are implemented for the computations. The area of the separation bubble and the enstrophy integrated over the separated region are calculated as indicators for the effectiveness of the synthetic jet on separation control. The more reduction in the area and enstrophy is obtained the better control is achieved. The selected parameters of the synthetic jet to manage improved control are $h_0$, $d_0$, $W$, $H$, $A$, $f$, and $x_{jet}$. The following observations are made based on the results:

- Synthetic jet actuator is found to have immense effect in controlling the separated region in terms of the separation bubble size and its shape.
- It is observed that in addition to the parameters, the effectiveness of the actuator highly depends on the momentum of the main flow to be controlled.

- The reduction in separated region size is observed to vary at different stages of the membrane oscillation cycle. The biggest reduction is achieved at the minimum volume stage for all the cases. Also, the separation bubble is totally eliminated in some cases.

- Maximum velocities (towards or outwards from the cavity) are observed to occur when the membrane is at its lowest as well as highest levels in contrast with the observation made for the quiescent external medium cases. This has been also confirmed with the reduction in the area and enstrophy of the separated region.

- The area of the separation is altered as a function of the jet location.

- The best result is obtained with relatively moderate value of the throat width, although the separation bubble is totally eliminated at all stages for a larger $d_o$ for which the second best case is obtained. A relatively worse result is obtained when the throat width has its minimum value.

- Smaller values of throat height do not produce as good results as the cases with larger throat heights. Also, increasing the throat height further from a certain value does not reduce the separation bubble, although more reduction in enstrophy is managed.

- The favorable effect is lessened as the cavity height is increased. Similar observation is made for the cavity width.
Better effectiveness is obtained as the membrane oscillation amplitude is increased. But, further increasing the amplitude from a certain value attenuates the favorable effect.

The membrane oscillation frequency is observed to improve the performance of the actuator. The percent reduction in separation area is steeper as compared to that in enstrophy.

Enlarging the synthetic jet cavity size does not create better results compared to the best case as one may intuitively expect. Furthermore, the enlargement followed by an increase in amplitude helps improve the performance of the actuator.

Finally, the synthetic jet actuator is introduced as an alternative way for thermal management of a microelectronic chip. For its demonstration, the microelectronic chip is placed on the bottom wall of a rectangular channel. Then, an actuator is positioned on top of the microelectronic chip directing the flow so as to remove the heat from the chip. Computations are carried out using slip flow regime models in order to predict the flow physics accurately. The following observations are deduced from the results obtained:

- It is shown that synthetic jet placement has a crucial role in thermal management; the best control is achieved when the synthetic jet is placed midway of the chip length.
- It is shown that more effective cooling is achieved as $h_o/d_o$ ratio of the throat is increased. Using a directed synthetic jet towards the channel exit does not secure a better thermal management. On the contrary, a higher
heat transfer rate from the microelectronic chip surface is achieved by using a non-directed synthetic jet with the same geometric and actuation parameters as the directed one. Also, using nozzle-like throat geometry is shown to help reduce the mean temperature over the microchip surface, thereby, increase the heat transfer rate from the microelectronic chip surface.

- Thermal management strengthens as the heat rate imposed on the microchip surface is increased.
- Using two synthetic jets improves the cooling provided that proper locations are designated for the synthetic jets.
- Phasing the membranes is also studied. It is showed that a 180 degree phase lag of the membrane oscillations between the two jets engenders better results in terms of heat transfer rates when compared to the zero degree of phase angle.
- Three synthetic jets are placed in tandem on the top wall of the microchip. Interesting to note that adding one more jet to the configuration does not yield much better results as initially expected. On the contrary, the results obtained by three jets are almost identical to those with one jet.
- According to the results obtained, the present approach can be suggested not only for overall thermal control of heated surfaces but also for thermal management of hot spots.

Results of the present study clearly show that the synthetic jet actuator proves itself to be an effective device for thermal-fluid control applications where low-speed
flows are encountered. It is suggested that the geometric and actuation parameters pertaining to the effectiveness of the actuator can be altered to attain desired control levels. The designs presented for separation control and thermal management of microelectronic chip are suggested to be compatible due to their flexibility in terms of sizing and configuration of the control systems.

The simulations in the present study are carried out for synthetic jets in two-dimensions. Therefore, it can be considered as a baseline study of a three-dimensional synthetic jet. As a continuation of the present study following suggestions can be made:

- As far as the separation control applications in internal flows are considered, the effect of the channel flow momentum on the synthetic jet performance should be further studied. A study can be conducted in search for finding a key parameter to identify effectiveness of synthetic jets on flow control in terms of both jet momentum and the momentum of the main flow to be controlled.

- As for thermal management of microelectronic devices, similar computations can be carried out for the same geometric and actuation designs but scaled for smaller scales. Also, a future study should present the limitations of the present design.

- Currently, a typical MEMS device is designed in a trial-and-error manner as it is done in the present study. This approach requires numerous iterations before the performance requirements of a given device are satisfied. However, a computational method can be coupled with an
automated design optimization methodology in order to eliminate long
time and high cost for commercial designs.
LITERATURE CITED


CURRICULUM VITA

DEGREES EARNED:

Doctor of Philosophy (Aerospace Engineering), Old Dominion University, Norfolk, Virginia, August 2003.

Master of Science (Aeronautical Engineering), Istanbul Technical University, Istanbul, Turkey, December 2001.

PROFESSIONAL CHRONOLOGY:

Old Dominion University, Norfolk, Virginia, Research Assistant, August 2002-present.

Istanbul Technical University, Istanbul, Turkey, Research Assistant, November 1999-May 2003.


SCHOLARY ACTIVITIES COMPLETED:

Journal publications:


Conference Papers with Proceedings:


Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.