Analysis and optimization of film cooling effectiveness

Hessam Babaee

Louisiana State University and Agricultural and Mechanical College

Follow this and additional works at: https://digitalcommons.lsu.edu/gradschool_dissertations

Part of the Mechanical Engineering Commons

Recommended Citation

https://digitalcommons.lsu.edu/gradschool_dissertations/2178

This Dissertation is brought to you for free and open access by the Graduate School at LSU Digital Commons. It has been accepted for inclusion in LSU Doctoral Dissertations by an authorized graduate school editor of LSU Digital Commons. For more information, please contact gradetd@lsu.edu.
A Dissertation

Submitted to the Graduate Faculty of the
Louisiana State University and
Agricultural and Mechanical College
in partial fulfillment of the
requirements for the degree of
Doctor of Philosophy

in

The Department of Mechanical Engineering

by
Hessam Babaee
M.Sc., University of Tehran, 2006
August 2013
Acknowledgments

First and foremost I would like to thank my adviser, Professor Sumanta Acharya, for his trust in me. This work has greatly benefited from his unparalleled vision in film cooling and his constant encouragement and support throughout my PhD. I also would like to thank my committee members Dimitris E. Nikitopoulos, Guoxiang Gu and Xiaoliang Wan. I would like to express my appreciation to Dr. Xiaoliang Wan, in particular, for many hours of insightful conversations we had together, that helped me gain insight about spectral methods.

I would also like to express my gratitude to Professor Vahid Esfahanian, my adviser in University of Tehran, who has been my first teacher in numerical methods. His high standards of performing research have been and will be my guidelines throughout my research.

This research was supported by a grant from AFOSR and a grant from the DEPSCoR-AFOSR programs. This support is gratefully acknowledged. The simulations were performed on Louisiana Optical Network Initiative (LONI) systems and High Performance Computing systems at Louisiana State University.

My friends made my journey as a graduate student a wonderful experience. The coffee breaks with Ataollah Mesgarnejad, Kasra Fattah and Luis Alvergue and our conversations about politics, religion, science and a lot of other subjects are of my most memorable moments. I particularly would like to thank Ali and Kaveh for their friendship which has enriched my life in unmeasurable ways.

Last and most of all I would like to thank my parents, Ahmad and Mahin, for their never-ending love. They have taught me the importance of education and have been supporting me every step of my life. Much of the credit of the completion of this dissertation should go to them.
# Table of Contents

ACKNOWLEDGMENTS ................................................................. ii

LIST OF TABLES ........................................................................ vi

LIST OF FIGURES ........................................................................ vii

ABSTRACT .................................................................................. xii

CHAPTER

1  INTRODUCTION ........................................................................ 1
   1.1  Film cooling ................................................................. 1
        1.1.1 Dynamics of a jet in a crossflow .............................. 1
        1.1.2 Optimization of film cooling effectiveness .................. 5
        1.1.3 Uncertainty quantification in film cooling applications ...... 7
   1.2  Finite difference on complex geometries ......................... 8
   1.3  Thesis overview ............................................................ 13
        1.3.1 Contributions ....................................................... 13
        1.3.2 Organization ....................................................... 14

2  NUMERICAL METHOD AND VALIDATION .................................. 16
   2.1  Mathematical model ...................................................... 16
   2.2  Spectral/hp element method .......................................... 17
        2.2.1 High-order splitting scheme .................................. 18
        2.2.2 Modal polynomial expansion ................................. 18
   2.3  Validation ................................................................. 20
        2.3.1 Vertical jet in a crossflow ...................................... 21
        2.3.2 Inclined jet in a crossflow .................................... 22

3  THE EFFECT OF PLENUM ON JET IN CROSSFLOW ................. 25
   3.1  Problem statement ....................................................... 25
   3.2  Numerical method ....................................................... 26
        3.2.1 Computational mesh ......................................... 26
   3.3  Results ................................................................. 28
        3.3.1 Flow inside the delivery tube ................................. 28
        3.3.2 Near and far field flow .................................... 31

4  OPTIMIZATION OF FORCING PARAMETERS OF FILM COOLING EFFECTIVENESS .................................................. 35
   4.1  Methodology ............................................................. 35
        4.1.1 Optimization problem ...................................... 37
        4.1.2 Discretization in space-time ................................. 38
        4.1.3 Discretization in design parameter space .................. 38
   4.2  Simulation details ....................................................... 42
4.2.1 Computational mesh ........................................ 42
4.2.2 Convergence in space and time .......................... 44
4.2.3 Convergence in design space ............................. 45
4.3 Results and discussions ........................................ 46
  4.3.1 Effect of pulsation on film cooling performance ...... 46
  4.3.2 Optimal and sub-optimal behavior ..................... 51
  4.3.3 Film cooling response surface ......................... 57
4.4 Conclusions .................................................... 58

5 UNCERTAINTY QUANTIFICATION OF FILM COOLING EFFECTIVENESS ........................................ 61
  5.1 Problem specification ........................................ 61
  5.2 Numerical algorithm ......................................... 65
    5.2.1 Stochastic discretization .............................. 66
    5.2.2 Discretization in space and time .................... 72
  5.3 Results ...................................................... 75
    5.3.1 Effect of blowing ratio on film cooling .............. 75
    5.3.2 Spatially-averaged film cooling effectiveness ...... 76
    5.3.3 Spanwise-averaged film cooling effectiveness ...... 82
  5.4 Concluding remarks ......................................... 86

6 A FINITE DIFFERENCE METHOD FOR SOLVING INCOMPRESSIBLE NAVIER-STOKES EQUATIONS .......................... 89
  6.1 Governing equations ......................................... 89
  6.2 Numerical algorithm ......................................... 90
    6.2.1 Metrics and discrete operators ....................... 92
    6.2.2 Provisional momentum equation ....................... 93
    6.2.3 Projection step ......................................... 95
    6.2.4 Summary of the algorithm .............................. 99
  6.3 Results ...................................................... 100
    6.3.1 Taylor-Green vortex .................................. 100
    6.3.2 Mixed convection in a lid-Driven cavity ............ 103
    6.3.3 Flow past cylinder .................................... 107
  6.4 Concluding remarks ......................................... 109

7 A SYMMETRIC DISCRETIZATION FOR PRESSURE-POISSON EQUATION .................................................. 112
  7.1 Introduction .................................................. 112
  7.2 Equations .................................................... 114
  7.3 Discretization ............................................... 116
    7.3.1 Discrete operators .................................... 117
    7.3.2 Discretization of Poisson-Neumann ................ 117
  7.4 Results ...................................................... 124
    7.4.1 Two-dimensional test case ............................ 124
## List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Weights for the stiffly-stable time integrator scheme (see [50], chapter 4.)</td>
<td>19</td>
</tr>
<tr>
<td>4.1</td>
<td>Grid convergence cases</td>
<td>44</td>
</tr>
<tr>
<td>4.2</td>
<td>Optimal design points</td>
<td>59</td>
</tr>
<tr>
<td>5.1</td>
<td>Multi-element decomposition of the random space</td>
<td>72</td>
</tr>
<tr>
<td>6.1</td>
<td>Discrete differential operators</td>
<td>93</td>
</tr>
<tr>
<td>6.2</td>
<td>List of stencil coefficients of equation 6.21 for semi-staggered and staggered grid</td>
<td>98</td>
</tr>
<tr>
<td>6.3</td>
<td>Grid convergence study for two-dimensional driven cavity</td>
<td>105</td>
</tr>
<tr>
<td>6.4</td>
<td>Comparison of Strouhal number and drag coefficient with measurements</td>
<td>108</td>
</tr>
<tr>
<td>7.1</td>
<td>Continuous and discrete differential operators</td>
<td>117</td>
</tr>
<tr>
<td>7.2</td>
<td>Grid convergence study for symmetric and non-symmetric discretizations of Poisson-Neumann problem on G1</td>
<td>128</td>
</tr>
<tr>
<td>7.3</td>
<td>Volumetric flow rate $Q$ at different pipe sections for relative residual tolerance of $\text{tol} = 1 \times 10^{-5}$</td>
<td>132</td>
</tr>
<tr>
<td>7.4</td>
<td>Volumetric flow rate $Q$ at different pipe sections for relative residual tolerance of $\text{tol} = 1 \times 10^{-9}$</td>
<td>132</td>
</tr>
</tbody>
</table>
## List of Figures

1.1 Schematic of a typical jet engine, from [6] chapter 14. ........................................... 2

1.2 Volume rendering of scalar field at velocity ratio $R = 2$. The result is obtained by performing DNS in the current research. The details of the computation are given in chapter 2. ................................. 4

1.3 Grid layouts (a) staggered grid; (b) collocated grid at vertex; (c) collocated grid at cell center; (d) semi-staggered grid. .............................. 12

2.1 Modal basis for one-dimensional element with $P = 4$. ......................... 20

2.2 Jet in cross-flow schematic: (a) $x_1 - x_2$ view, (b) $x_1 - x_3$ view. .................. 22

2.3 Three-dimensional view of the hexahedral grid of the entire computational domain. ................................................................. 23

2.4 Instantaneous temperature at mid-plane ($x_3 = 0$) at velocity ratio $R = 2$. ................................................................. 24

2.5 Comparison of time-averaged $u_1$ velocity in the mid-plane with experimental data [13] at $BR = 0.15$. ...................................................... 24

3.1 Schematic of the jet in the crossflow: (a) $x_1 - x_2$ view; (b) $x_1 - x_3$ view. .......... 27

3.2 Unstructured hexahedral grid; (a) three-dimensional view; (b) $x_1 - x_3$ view of the grid in the vicinity of the jet exit with spectral order $m = 4$. ................................................................. 28

3.3 Flow conditions inside the delivery tube for velocity ratio $R = 0.4$: (a) projected streamlines and velocity vectors on the inlet of the delivery tube at $x_2 = -2$; (b) authors’ representation of flow inside the delivery tube; (c) velocity profile at mid-section $x_2 = -1$. The cross section is perpendicular to the pipe axis. Also the iso-surface of $\lambda_2 = -0.5$ is shown whose color is in gray scale with black (negative axial vorticity) to white (positive axial vorticity). . . 30

3.4 Time-averaged vertical velocity across the delivery tube at $R = 0.4$ at different cross sections. ................................................................. 31

3.5 Volume rendering of velocity magnitudes at velocity ratio $R = 0.4$ from $x_1 - x_2$ view. ................................................................. 31

3.6 Volume rendering of velocity magnitudes at velocity ratio $R = 0.6$ from $x_1 - x_2$ view. ................................................................. 32
3.7 Close up view of $\lambda_2$ iso-surfaces in the delivery tube at $R = 0.6$ for inclined jet in crossflow with a plenum. The gray scale is used black (negative streamwise vorticity) to white (positive streamwise vorticity). ................................................... 32

3.8 Volume rendering of scalar field at velocity ratio $R = 0.4$ for inclined jet in crossflow with a plenum. ................................................... 33

3.9 Volume rendering of scalar field at velocity ratio $R = 0.6$ for inclined jet with a plenum. ................................................... 34

4.1 Schematic of the jet in crossflow. ................................. 36

4.2 Blowing ratio signal $g(t; \xi)$ ........................................... 37

4.3 Unstructured hexahedral grid; (a) three-dimensional view; (b) $x_1 - x_3$ view of the grid in the vicinity of the jet exit with spectral order $m = 4$. ................................................... 43

4.4 Grid convergence study for center-line film and spanwise averaged cooling effectiveness. ........................................... 44

4.5 Error convergence for $\tilde{\eta}_N(\xi)$ with increasing polynomial order ........................................... 46

4.6 Time-averaged temperature contours for quadrature points on cooled surface ($x_2 = 0$). Each row: constant $T_p$; Each column: constant $DC$. ................................................... 47

4.7 Instantaneous temperature surface in the mid-plane ($x_3 = 0$) with constant pulsation period of $T_p = 1.16$. ........................................... 49

4.8 Instantaneous temperature surface in the mid-plane ($x_3 = 0$) with constant duty cycle of $DC = 0.52$. ........................................... 50

4.9 Snapshots of instantaneous temperature at $x_3 = 0$ during one pulsation cycle with $DC = 0.09$, $T_p = 1.16$, $\Delta t_{on} = 0.10$ and $\Delta t_{off} = 1.06$. ................................................... 52

4.10 Instantaneous $u_2 - u_3$ velocity vector field at: left $x_1 = 2$; right $x_1 = 4$, at $DC = 0.09$ and $T_p = 1.16$. ........................................... 54

4.11 Snapshots of instantaneous temperature at $x_3 = 0$ during one pulsation cycle with $DC = 0.34$, $T_p = 1.16$, $\Delta t_{on} = 0.39$ and $\Delta t_{off} = 0.77$. ................................................... 55
4.12 Snapshots of instantaneous temperature at $x_3 = 0$ during one pulsation cycle with $DC = 0.09$, $T_p = 2.75$, $\Delta t_{on} = 0.25$ and $\Delta t_{off} = 2.5$.

4.13 Contours of averaged film cooling effectiveness ($\bar{\eta}_N(\xi)$). Local and global maxima are shown, with $P_1$ being the global maximum and $P_2$ to $P_5$ are local maxima ordered with decreasing value.

5.1 Schematic of jet in a crossflow.

5.2 Probability density function of blowing ratio; a truncated Gaussian distribution with mean of 0.3 and variance of 0.01. Elemental decomposition ($B_e$, $e = 1, \ldots, Ne$) is shown schematically.

5.3 Polynomial basis $\phi^e(\xi)$; top: element $B_1$; bottom: element $B_4$.

5.4 Unstructured hexahedral grid; (a) three-dimensional view; (b) $x_1 - x_3$ view of the grid in the vicinity of the jet exit, black lines: element boundaries; gray lines: Gauss-Lobatto-Legendre quadrature grid with spectral order $m = 4$.

5.5 Spectrum of the energy of the velocity signal at location $x_1 = 2$, $x_2 = 1$ and $x_3 = 0$ with blowing ratio of $BR = 0.5841$.

5.6 Instantaneous temperature surface in the mid-plane ($x_3 = 0$) for different blowing ratios.

5.7 Time-averaged temperature contours for quadrature points on cooled surface ($x_2 = 0$). In this figure simulations for all Gauss quadrature points are shown.

5.8 Spatially-averaged film cooling effectiveness $\bar{\eta}(\xi)$ for different projection order $M = 0, \ldots, 3$.

5.9 Probability density function for $\xi$, a comparison between sampled PDF and exact PDF.

5.10 Probability density function for film cooling effectiveness $\bar{\eta}(\xi)$.

5.11 Spanwise-averaged film cooling effectiveness $\bar{\eta}(x_1; \xi)$ versus different random blowing ratio at $x_1 = 2$, $3$, $4$, $6$, $8$, and $10$.

5.12 PDF of spanwise-averaged film cooling effectiveness at $x_1 = 2$, $3$, $4$, $6$, $8$, and $10$. Note that the horizontal axis in (a)-(f) corresponds to the vertical axis in figure 5.11.
5.13 Uncertainty in the spanwise-averaged film cooling effectiveness

\[ \eta(x_1; \xi) \] ................................................................. 85

5.14 Standard deviation of temperature, \( \sigma_\theta(x) \) on the cooled surface of \( x_2 = 0 \). ................................................................. 85

5.15 Sensitivity and standard deviation of spanwise-averaged film cooling effectiveness \( \eta(x_1; \xi) \). ................................................................. 87

6.1 (a) Grid cell in the computational space; (b) Indexing convention in the computational cell. \( \times \): face center, sphere: cell center. ......................... 91

6.2 Pressure-Poisson discretization on uniform Cartesian grid for (a) semi-staggered grid, (b) staggered grid ........................................... 97

6.3 Schematic of the projection step algorithm (a) interpolating \( \tilde{U}^i \) to its corresponding face center; (b) constructing \( \mathcal{DG}^i \) and solving pressure-Poisson problem; (c) updating the intermediate velocity \( \tilde{U} \) to divergence-free velocity field \( U^{n+1} \) located at the face center; (d) interpolating \( U^{n+1} \) from cell face center to cell vertex. .............. 101

6.4 65 \times 65 curvilinear grid with \( a = 0.1 \) and \( \omega = 10\pi \): the entire computational domain (left); a closeup of interior grid (top right); a closeup of grid near boundary (bottom right). ..................... 102

6.5 (a) and (b) Horizontal velocity contours. Dashed: hybrid semi-staggered grid (HSSG); solid: exact solution; (a) 33 \times 33 grid points; (b) 65 \times 65 grid points; (c) spatial convergence for Taylor-Green vortex problem. ......................................................... 104

6.6 Lid-driven cavity at \( Re = 100 \) and \( Gr = 0 \): (a) horizontal velocity in the vertical mid-plane (b) contours of \( \phi \) in the range of \((-0.5, 0.5)\) with increment of 0.05. ........................................... 105

6.7 Lid-driven cavity: (a) and (b) \( Re = 400 \) and \( Gr = 10^4 \); (a) \( \theta \) at \( x_1 = 0.5 \); (b) Temperature filled contours; (c) and (d) \( Re = 400 \) and \( Gr = 10^6 \); (c) \( \theta \) at \( x_1 = 0.5 \); (d) Temperature filled contours. ................. 106

6.8 Multi-block H-grid generated around the cylinder. A total of 161 points are uniformly distributed around the cylinder and a total of 290 and 100 points are used in the \( x_1 \) and \( x_2 \) directions. ............... 108

6.9 Flow over cylinder: (a) drag coefficient \( (C_d) \) at \( Re = 150 \); (b) lift coefficient \( (C_l) \) at \( Re = 150 \); (c) time-averaged pressure and streamlines for \( Re = 100 \); (d) mean separation angle \( \theta \) for flow over cylinder with \( \theta \) measured from the stagnation point. ................. 110
7.1  (a) 2D Grid cell in the physical space; (b) 2D indexing convention in the computational cell; (c) 3D Grid cell in the physical space; (d) 3D indexing convention in the computational cell................. 116

7.2  Schematic of an $\xi^1$-constant boundary with $i = 1$. ................................. 121

7.3  Test grids for discretization of Poisson-Neumann problem (a) G1: highly curvilinear grid ($41 \times 41$) (b) G2: rough grid ($41 \times 41$). ............... 125

7.4  Surface plot for grid G2 (a) $B^{11}$; (b) $B^{12}$; (c) $B^{22}$. ................................. 126

7.5  Contour lines of $\phi$ for (a) and (c) symmetric discretization; (b) and (d) non-symmetric discretization. ................................. 127

7.6  Error convergence for symmetric and non-symmetric discretization on G2. ................................................................. 128

7.7  Satisfaction of the discrete continuity $D^\xi(U)$ for (a) G1; (b) G2. ............... 129

7.8  (a) Plane view schematic of the $90^\circ$-bend; (b) $61 \times 51 \times 51$ curvilinear grid. ................................................................. 130

7.9  Comparison between Reynolds number for the pulsatile flow in the pipe calculated from analytical solution and experimental measurements. ................................................................. 131

7.10 Streamwise velocity at symmetry plane at time $t = T/2$. ................................. 134

7.11 Streamwise velocity at symmetry plane at time $t = T$. ................................. 135
Abstract

In the first part, an optimization strategy is described that combines high-fidelity simulations with response surface construction, and is applied to pulsed film cooling for turbine blades. The response surface is constructed for the film cooling effectiveness as a function of duty cycle, in the range of $DC$ between 0.05 and 1, and pulsation frequency $St$ in the range of 0.2-2, using a pseudo-spectral projection method. The jet is fully modulated and the blowing ratio, when the jet is on, is 1.5 in all cases. Overall 73 direct numerical simulations (DNS) using spectral element method were performed to sample the film cooling effectiveness on a Clenshaw-Curtis grid in the design space. It is observed that in the parameter space explored a global optimum exists, and in the present study, the best film cooling effectiveness is found at $DC = 0.14$ and $St = 1.03$. In the same range of $DC$ and $St$, four other local optimums were found. The gradient-based optimization algorithms are argued to be unsuitable for the current problem due to the non-convexity of the objective function.

In the second part, the effect of randomness of blowing ratio on film cooling performance is investigated by combining direct numerical simulations with a stochastic collocation approach. The blowing ratio variations are assumed to have a truncated Gaussian distribution with mean of 0.3 and the standard variation of approximately 0.1. The parametric space is discretized using Multi-Element general Polynomial Chaos (ME-gPC) with five elements where general polynomial chaos of order 3 is used in each element. Direct numerical simulations were carried out using spectral/hp element method to sample the governing equations in space and time. The probability density function of the film cooling effectiveness was obtained and the standard deviation of the adiabatic film cooling effectiveness on the blade surface was calculated. A maximum standard deviation of 15% was observed in the region within a four-jet-diameter distance downstream of the exit hole. The spatially-averaged adiabatic film cooling effectiveness was $0.23 \pm 0.02$. The calculation of all the statistical properties were carried out as off-line post-processing. Overall the computational strategy
is shown to be very effective with the total computational cost being equivalent to solving twenty independent direct numerical simulations that are performed concurrently.

In the third part, an accurate and efficient finite difference method for solving the incompressible Navier-Stokes equations on curvilinear grids is developed. This method combines the favorable features of the staggered grid and semi-staggered grid approaches. A novel symmetric finite difference discretization of the Poisson-Neumann problem on curvilinear grids is also presented. The validity of the method is demonstrated on four benchmark problems. The Taylor-Green vortex problem is solved on a curvilinear grid with highly skewed cells and a second-order convergence in $l_{\infty}$-norm is observed. The mixed convection in a lid-driven cavity is solved on a highly curvilinear grid and excellent agreement with literature is obtained. The results for flow past a cylinder are compared with the existing experimental data in the literature. As the fourth case, three dimensional time-dependent incompressible flow in a curved tube is solved. The predictions agree well with the measured data, and validate the approach used.
Chapter 1
Introduction

Gas turbines are widely used in land-based power plants and aircraft jet engines. The main components of a typical gas turbine include compressor, combustion chamber and turbine. In compressor the pressure of the process fluid increases from ambient pressure which is then mixed with fuel in the combustion chamber (see figure 1.1). The ignition of the high pressure mixture of gas and fuel increases the energy level of the fluid. In the next stage, the mixture with high pressure and high temperature and at its highest level of energy during the cycle, is directed through nozzles over turbine’s blades where its energy is extracted. The converted energy is used to provide power to compressor and it also drives power generator in the case of land-based gas turbines.

1.1 Film cooling

To enhance the thermal efficiency of a gas turbine, higher rotor inlet temperatures are desired requiring more effective cooling strategies of the components downstream. Film cooling of the high pressure turbine is commonly employed in modern turbine designs [14]. The coolant flow is extracted from the the compressor and is bled through discrete film holes on the surface of the blade. The role of the coolant layer is to protect the components on the hot gas path and therefore increases the life of these components. The interaction between the coolant air and the mainstream causes aerodynamic losses in the turbine stage. In addition, coolant air represents a loss of the process air available for power or thrust. For these reasons one of the main objectives in cooling design is to use coolant as minimally as possible while ensuring a proper coverage of the coolant on the hot gas path components.

1.1.1 Dynamics of a jet in a crossflow

Understanding the dynamics of jet in crossflow is crucial in several applications such as film cooling of gas turbines, thrust vectoring of high speed turbojets and VSTOL aircrafts, pollutant dispersal from chimneys and fuel injection in combustion chambers. There have been extensive experimental [30, 51, 86, 11] and numerical [93, 65, 80, 64] studies that have
explored the dynamics of jet in crossflow. Five vortical structures have been found that dominate the dynamic of the jet in crossflow:

1. The Counter-rotating Vortex Pair (CVP) which is a pair of quasi-streamwise vortices and is largely responsible for mixing of the jet and crossflow. Experimental observations by Kelso et al. [51] for the velocity ratios, denoted by $R = U_j/U_\infty$; in the range of $2 \leq R \leq 6$ show that CVP is originated by tilting and folding of the vortex rings that are created from the periodic rollup of the shear layer vortices. These observations were later confirmed by the computational study carried out by Corteletti & Karagozian [20].

2. The shear layer vortices as shown in figure 1.2 (see next chapter for more explanation about figure 1.2). The shear layer vortices are commonly accepted to be the result of Kelvin-Helmholtz instability of the cylindrical vortex sheet that emerges from the jet nozzle [8]. Recently the experimental study by Davitian et al. [22] showed that the vertical jet in crossflow is globally unstable for velocity ratios below the range of 3.0-3.3 and is convectively unstable for the velocity ratios above this range.

3. Horseshoe vortex which appears at the upstream side of the jet as shown in figure 1.2. Figure 1.2 shows the volume rendering of the scalar field at velocity ratio $R = 2$. (The
result shown in figure 1.2 are obtained by performing direct numerical simulation with the same method that is used throughout this study and more details are given in chapter 2.) This vortex is formed due to the separation of the crossflow boundary layer on the upstream side of the jet. The boundary layer separation is caused by the adverse pressure gradient resulted by the obstruction of crossflow by the transverse jet, which creates a spanwise vortex that wraps around the jet body and forms the horseshoe vortex (see figure 1.2).

4. The upright vortices which connect the jet body to the wall vortices as shown in figure 1.2. These quasi-vertical vortices appear in the jet wake and their formation, as suggested by Kelso et al., Fric & Roshko [51] and [30], is caused by the separation of the crossflow behind the jet column which, for sufficiently large velocity ratios, serves as a “soft body”. These vortices shed with low frequency which is in the same order of magnitude as the von-Kármán vortex street for flow over a cylinder with the crossflow velocity $U_\infty$ and a cylinder with the diameter $D$ Schlatter et al. [83].

5. The wall vortices which are a quasi-streamwise pair of counter-rotating vortices which have steady presence close to the wall surface.

The majority of the literature including all references mentioned above have focused on jet in crossflow with high velocity ratios. At low velocity ratios the vortical structures of jet in crossflow are very different as revealed in the experimental study carried out by Acarlar & Smith [2] who considered the injection through surface slots in a flat plate in the interest of investigating the near-wall vortical structures present in turbulent boundary layer. They observed that hairpin vortices are the dominant vortical structures at low-velocity-ratio jets in crossflow. Haven & Kurosaka [41] carried out experiments for velocity ratios in the range of $0.4 \leq R \leq 2.0$ in which the effect of six different hole shapes on the near-field dynamics of jet in crossflow were investigated. Gopalan et al. [34] studied vertical jet in crossflow for velocity ratios in the range of $0.5 \leq R \leq 2.5$. At low velocity ratios they observed a
semi-cylindrical vortical layer forming behind the jet. They observed a low velocity region under the jet and between the legs of the semi-cylindrical vortical structure. Recently Bidan & Nikitopoulos[11] studied steady and pulsed vertical jet in crossflow with velocity ratio in the range of $0.15 \leq R \leq 4.2$. At low velocity ratios they observed shedding of hairpin vortices and presence of streamwise ("side") vortices downstream of the jet exit.

Inclined jet in crossflow at low velocity ratios has been the subject of several experimental and numerical studies with the focus of film cooling applications (see [14] and references therein) where inclining the coolant jet in the symmetry plane is an attempt to prevent the jet liftoff from the surface and thus preventing the penetration of crossflow with high temperatures to the wake of the jet. In practical configurations, the inclined jet is attached to a supply plenum. Although it has long been known from the experimental observations [70] that entrance of the flow from the plenum into the delivery tube can significantly affect the flow characteristics at the jet exit, there are very few studies that have explored the impact of plenum in near-field and far-field jet in crossflow. Although the plenum has been
included in several numerical models [57, 44, 81, 63, 3], the dynamics of the flow inside the delivery tube and its impact on flow characteristics are not well-understood. In chapter 3, we closely investigate the flow inside the delivery tube for inclined jet attached to a plenum. We perform two DNS simulations at low velocity ratio of $R = 0.4$ and $R = 0.6$.

1.1.2 Optimization of film cooling effectiveness

In order to improve cooling effectiveness, there have been a large number of studies that have explored hole shapes, blowing ratios, coolant injection angles and other design parameters [17, 58, 14]. The majority of the studies that have explored parametric effects are either experimental or simulations that utilize the Reynolds-Averaged Navier-Stokes (RANS) equations and a turbulence model. However, turbulence models in RANS have been shown to not capture the dynamics of the flow field accurately, and high-fidelity spatially and temporally resolved simulations that do not require models to capture the effects of the entire turbulent spectrum are needed [5]. Such techniques represented by Direct Numerical Simulations (DNS), which resolves the entire spectrum, or Large Eddy Simulations (LES), that captures the energetic parts of the spectrum, have indeed been recently employed in a number of film cooling studies [4, 45, 66, 37, 73]. Such high-fidelity simulations are used in the present work.

In recent years, pulsation of the coolant air has been suggested as one of the strategies that can possibly increase the film cooling effectiveness. Several researchers have studied the impact of pulsation on film cooling effectiveness, but the reported results have been mixed with no clear conclusive outcomes. Some key studies are reviewed below in order to motivate the present work.

Coulthard et al. [21] carried out an experimental study for an inclined jet with inclination angle of 35 degree. Their results show that the steady jet with BR, the ratio of the jet velocity to the crossflow velocity, of 0.5 resulted in the best film cooling effectiveness, and pulsations adversely impacted cooling effectiveness. Ekkad et al. [24] performed an experimental study in which they investigated the effect of pulsation on the film cooling
effectiveness on a leading edge model for a variety of forcing parameters. At the lowest blowing ratio, BR=0.75, the effect of pulsation was negligible. However at higher blowing ratios pulsation had a positive effect on the film cooling effectiveness. They also observed that the pulsed jet outperformed the steady jet with the same time-averaged blowing ratio. El-Gabry and Rivir[25] studied the effect of pulsation on the leading edge film cooling on the same geometry[24], and observed that the film effectiveness of the pulsed cases were lower than that of steady jets with the same time-averaged blowing ratios.

Muldoon and Acharya [63] performed direct numerical simulations for St in the range of 0.004 to 0.32 and $DC$ of 0.25 and 0.5. The baseline simulation was a steady jet at BR=1.5. Pulsed cases at $DC = 0.5$ improved the film cooling effectiveness compared to the baseline case. The pulsed jet also showed improved film cooling effectiveness than a steady jet with the same effective mass flowrate, i.e. BR=0.75.

Recently Bidan et al. [12] carried out an experimental/numerical study of a 35-degree jet in a crossflow. They observed that pulsation resulted in decrease of film cooling performance when compared against steady jets at the same mass flow rate. However, pulsated jets showed improvement of film cooling effectiveness when compared with steady cases with the same high blowing ratio.

It is clear from the studies above [21, 24, 25, 63] that the reported results are not consistent with both positive and negative effects of coolant pulsations on film cooling. Further, in the experimental studies reported in the literature, there is a significant amount of deviation between the desired instantaneous blowing ratio and the actual delivered pulse form. This variability is usually more pronounced at higher frequencies. For instance, experimental measurements of [21] shows 100% deviation between the phased average blowing ratio at $St = 0.19$ compared to the desired blowing ratio. In the experimental studies carried out by [25] and [12], significant amount of uncertainty, particularly at high $St$ in [12], is reported. This variability in delivering the desired blowing ratio for the pulsed film cooling, however unavoidable, renders the task of comparing these studies cumbersome if
not impossible, and this may, among other factors, explain the mixed results in the literature. Moreover the majority of the studies to date have focused on pre-selecting the forcing frequency based on available hardware or on natural time scales in the flow. Thus the parametric window exploring the role of pulsations have been limited, and it is possible that the optimal forcing frequency may be outside the pre-selected ranges considered.

1.1.3 Uncertainty quantification in film cooling applications

As a result of variability in operating conditions, the performance of film cooling can be unfavorably affected leading to the exposure of the hot-gas components to high temperatures beyond the permissible limit. This has severe impact on the gas turbine durability where increasing the airfoil temperature by $25^\circ C$ can reduce the life time of a blade by a factor of two [14]. The cost of the replacement of vane/blade airfoils is high, most of which is attributed to the manufacture or repair of the film cooling holes.

Film cooling effectiveness, among other factors, is strongly dependent on the blowing ratio (the ratio of the averaged coolant velocity in the delivery tube to the crossflow velocity as demonstrated by several experimental and numerical studies (see for example [9, 13]). The coolant film with very low blowing ratio ($BR < 0.1$) can be rapidly mixed out with the crossflow, which leads to the deterioration of the film cooling performance. Higher blowing ratios ($BR > 1$), on the other hand, cause the jet to lift off from the surface, which allows the hot gas from the crossflow to penetrate into the separated region downstream of the jet, resulting in a poor coverage of the blade surface. Intuitively, a blowing ratio exists which results in the best film cooling effectiveness, and the existence of an optimal blowing ratio that maximizes film cooling for specific operating conditions and geometry (typically a flat plate) has been reported in the literature. The optimum blowing ratio can vary depending on the operating conditions and geometry, and designers use empirical data and experience in designing the set-points for the film cooling parameters.

In practice, the blowing ratio is far from its designed value. Abhari [1] reported the coolant fluctuations of up to $\pm 100\%$ around the design blowing ratio due to the rotor-stator
interaction. Experimental measurements carried out by Womack et al. [91] showed that the blade heat transfer coefficient was significantly affected by the presence of wakes in the crossflow; this variation is, in part, due to variations in the blowing ratio. Thus, it is important to investigate the effect of blowing ratio variation on the cooling effectiveness.

In the current research we present a probabilistic framework to study the effect of randomness of design parameters on film cooling effectiveness.

1.2 Finite difference on complex geometries

In this dissertation, an improved finite difference algorithm to solve the time-dependent incompressible Navier-Stokes equations is presented on general curvilinear grids. Considering the incompressible Navier-Stokes equations are given by:

\[
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \nu \nabla^2 \mathbf{u},
\]

\[
\nabla \cdot \mathbf{u} = 0,
\]

where \( \mathbf{u} = \{u_1, u_2, u_3\} \) are the Cartesian velocity vector, \( p \) the pressure divided by density, and \( \nu \) is the kinematic viscosity. The incompressible Navier-Stokes equations 1.1-1.2 represent an evolution equation for the velocity field \( \mathbf{u} \) defined in a divergence-free vector space [28]. The continuity equation does not involve pressure, which is at the core of the difficulties encountered in the discrete representations of equations 1.1-1.2. The incompressibility constraint casts pressure, in a mathematical sense, as the Lagrange multiplier that ensures a divergence-free velocity field rather than a thermodynamic variable [35]. A pressure-Poisson equation is generally derived from equations 1.1-1.2, for which Neumann boundary condition is prescribed. Therefore, the pressure solution is not unique and is arbitrary up to an additive constant.

The proper treatment of the pressure-velocity coupling, in the finite difference/volume discrete representation of equations 1.1-1.2, has primarily motivated the idea of different grid structures, where different layout arrangements of velocity vector components and
pressure are employed. Different grid layouts result in different discrete operators whose properties determine the interrelation of velocity and pressure. We briefly overview the approaches using different grid layouts and discuss their characteristics before discussing the specific approach proposed in this paper.

Harlow and Welsh [40] used the staggered grid layout to solve the two-dimensional time-dependent incompressible flow on a Cartesian grid using the finite difference method. In the staggered grid layout, pressure is stored at the center of grid cells and velocity components at the center of the grid cell faces with the horizontal velocity located at the center of the vertical face and the vertical velocity at the center of the horizontal face (figure 1.3(a)). In this method, momentum equation is discretized for each velocity component at its corresponding face center. The velocity component of the other face is interpolated to the face for which the momentum equation is discretized. Taking advantage of the uniform Cartesian grid, they used ghost cells to enforce boundary conditions for the velocity components parallel to the physical boundaries. Perhaps the most favorable property of the staggered grid method is the fact that the discretization of the governing equations 1.1-1.2 does not exhibit any spurious pressure eigen-mode. Moreover, since pressure is located at the cell center, *ad hoc* pressure boundary conditions are not required to compute the pressure. The Neumann boundary condition is naturally enforced by incorporating the given normal pressure gradient into the discretization of the Laplacian of the pressure at cells adjacent to the boundary. This treatment of the Neumann boundary condition is shown to preserve the compatibility condition that is required to solve the singular Poisson-Neumann problem [52].

Since its introduction, the staggered grid has been extensively used to solve the incompressible Navier-Stokes equations using finite difference and/or finite volume methods. Most notably, Kim and Moin [52] used the staggered grid in conjunction with the projection method introduced by Chorin [18] to provide an efficient strategy to solve the time-dependent Navier-Stokes equations on Cartesian grids. The conservation properties of the
staggered grid approach [67, 61, 38] make it a particularly suitable method for direct and large eddy simulations of the turbulent flows where “physical accuracy” is strictly required [60, 95].

In extending the staggered grid arrangement to general curvilinear grids, the common approach is to map the grid from the physical space to a uniform grid in the computational space. The governing equations are then correspondingly mapped to the computational space by transforming the Cartesian coordinates to the curvilinear coordinates. The dependent variables can be either the Cartesian velocity or the contra-variant velocity components. In geometries such as a 90° bend, there are cells in which the Cartesian velocity components become tangent to their corresponding faces. This leads to an odd-even decoupling in the pressure if the Cartesian velocity components are the dependent variables [84]. This drawback precludes this method from being applied to general complex geometries. An alternative to this approach, employed by Maliska and Raithby [59], is to solve for all three Cartesian velocity components of the momentum equations at each face. Although accurate, this method effectively triples the cost of the computation which in three dimensional time-dependent simulations can be computationally inefficient. In using the contra-variant velocity as the dependent variable (see for example [53, 96]), the appearance of the Christoffel symbol in the transformed equations demands the mapping to be at least twice continuously differentiable. This restriction on the mapping is difficult to achieve for many complicated geometries. Furthermore computing or storing the Christoffel symbol is computationally expensive. Rosenfeld et al. [79, 78, 77], instead, projected the momentum equation into face-normal directions, thereby obtained an evolution equation for the contra-variant volume flux. Other than avoiding the interpolation of Cartesian velocity across the cell faces, in their method the Christoffel symbol is not used. This advantage, in fact, relaxes the constraint on the smoothness of the mapping. They successfully solved the time-dependent Navier-Stokes equations in a three dimensional domain. However, in their method, to project the Navier-Stokes equation to the face-normal direction, all three
components of the momentum equations have to be evaluated at each face center. This essentially triples the computational cost of advancing the momentum equations.

Another challenge in extending the staggered approach to general curvilinear grids is the issue of boundary condition enforcement for the velocity components that are parallel to the physical boundary. In the finite difference method, the spatial differential operators must be discretized at points adjacent to the physical boundary while incorporating the boundary condition. Using ghost cells in general curvilinear grids is complicated if not impossible (particularly if several ghost layers are needed due to the wider stencil width of the higher order schemes). The difficulty of utilizing ghost cells arises from the need to evaluate the metric terms in the ghost cells. This would require the coordinates of the ghost cells to be first created from which the metric terms are to be calculated. Such a ghost layer must, however, form a smooth extension of the interior grid coordinates to ensure the accuracy of the discretization at near-boundary-regions, where accuracy is often most required. Furthermore the ghost layer grid lines must not fold or overlap. To our knowledge, a systematic procedure to create several ghost layers that guarantees the above mentioned restrictions has not yet been proposed.

Perhaps the most seemingly natural choice for the grid layout is the collocated (or non-staggered) approach where velocity components and pressure are stored at the same location, either at the vertex of the cell (figure 1.3(b)) or at the center of the cell (figure 1.3(c)). It is a well-known fact that the discrete representation of velocity and pressure on collocated grid is not immune to spurious pressure modes. To suppress the spurious pressure eigen-modes, Rhie and Chow [74] used regularizing terms in the continuity equation which results in a proper coupling between velocity and pressure. Interpolation of velocity components across different grids is naturally avoided and velocity boundary conditions are directly enforced at the physical boundaries. These favorable features have contributed to the ease of the extensibility of the collocated grid to the curvilinear grid (see for example [55, 62]). However due to the fourth-order artificial terms that are added to the continuity
Figure 1.3: Grid layouts (a) staggered grid; (b) collocated grid at vertex; (c) collocated grid at cell center; (d) semi-staggered grid.

equation, the discrete conservation of mass is violated. This loss of physical accuracy becomes particularly important, for example, in direct and large eddy simulations of turbulent flows.

Recently, Ge et al. [31] proposed a numerical procedure based on the collocated approach in which the discrete continuity is satisfied exactly. In their method, velocity and pressure are stored at cell vertices (figure 1.3(b)). To construct the divergence operator, however, the interpolated contra-variant flux components at the face centers were used instead of using the velocity vector at the vertex. This results in proper coupling between pressure and velocity, and the exact satisfaction of discrete continuity. However, similar to any vertex-collocated approach (figure 1.3(b)), ad hoc pressure boundary conditions must be enforced at the cell vertices on the physical boundaries.
Another grid layout, which compared with the staggered and collocated grid arrangements has received little attention in the literature, is the semi-staggered approach where the velocity is stored at the cell vertex and pressure is located at the cell center (figure 1.3(d)). Fortin et al. [29] and Kuznetsov [54] employed the semi-staggered to solve the incompressible Navier-Stokes equations. In semi-staggered grid, all components of the momentum equation are discretized at cell vertices. Furthermore, since pressure is stored at the cell center, ad hoc boundary conditions are not required to solve for pressure, and the discrete continuity is satisfied exactly. This is an important advantage similar to the staggered grid. However, semi-staggered grid layout is not immune to spurious pressure modes [29, 69]. Ragab et al. [48] successfully used the semi-staggered layout on an orthogonal grid to solve the time-dependent incompressible Navier-Stokes equations. To avoid the spurious pressure modes they calculated the pressure-gradient at the cell face rather than the cell vertex. In a recent work by Figueiredo et al. [27], an investigation of the accuracy of different grid layouts was carried out. They solved the incompressible Navier-Stokes equations on a Cartesian grid with all four different grid layouts shown in figure 1.3. To remove the checker-boarding oscillations from the pressure, after solving the pressure-Poisson equation, they interpolated pressure across all the neighbor cells and obtained a smooth pressure field. In their comparison, the semi-staggered grid approach obtained the most accurate results for a regularized cavity problem.

1.3 Thesis overview

1.3.1 Contributions

This main contribution of the current research is the use of high-order spectral numerical method for solving film cooling application. The computational advantage of high order accuracy has allowed efficient discretization in space and time and also the discretization of the design space. As a result, computationally intensive heat transfer problems such as optimization of film cooling effectiveness and uncertainty quantification of film cooling effectiveness have been carried out with fast-converging spectral methods. More specifically
the contributions of the current research are as follows:

- **Film cooling**
  1. A novel optimization strategy is proposed to optimize film cooling effectiveness.
  2. This is the first study that proposes a probabilistic framework for investigating the effect of uncertainty in design parameters on film cooling effectiveness.

- **Finite difference**
  1. A novel finite difference algorithm is proposed to solve time-dependent incompressible Navier-Stokes equations.
  2. A novel symmetric finite difference discretization for solving pressure-Poisson equation on curvilinear grid is suggested.

1.3.2 **Organization**

The current research has focused on two separate topics; (1) optimization and uncertainty quantification of film cooling effectiveness presented in chapters 2-5, and (2) the development of finite difference method for incompressible Navier-Stokes equations that is discussed in chapters 6 and 7.

In chapter 2, the governing equations along with their discretization in space and time are demonstrated. The validation for the numerical approach is also given. In chapter 3, the effect of plenum on a jet in a crossflow is investigated. In chapter 4, an optimization strategy to maximize film cooling effectiveness with duty cycle and pulsation period as design parameters is proposed. In chapter 5, a probabilistic framework that was presented in detail in [7] is used to quantify the effect of uncertainty in blowing ratio on film cooling effectiveness.

In chapters 6 and 7, a finite difference algorithm to solve time-dependent incompressible Navier-Stokes equations is presented.

Chapters 3-7 are written in a self-contained fashion. Each of these chapters are taken
from a separate journal/conference paper and have been reproduced here with slight modifications.
Chapter 2
Numerical Method and Validation

2.1 Mathematical model

To study the jet in a crossflow (JICF) in film cooling applications, we make the following assumptions to reach at the mathematical model that governs the flow:

1. Flow is Newtonian.

2. Flow is incompressible.

3. The density of the jet and crossflow fluid are the same i.e iso-density JICF.

4. Kinematic viscosity and thermal diffusivity are temperature-independent and are assumed to be constant.

5. Buoyancy is neglected.

Once those assumptions are made, the flow is governed by incompressible Navier-Stokes equation along with the energy equation in the following form:

\[ \nabla \cdot \mathbf{u} = 0, \quad (2.1) \]

\[ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u}, \quad (2.2) \]

\[ \frac{\partial \theta}{\partial t} + (\mathbf{u} \cdot \nabla)\theta = \frac{1}{RePr} \nabla^2 \theta. \quad (2.3) \]

In the above equations, \( \mathbf{u} = \mathbf{u}(x,t) \), \( p = p(x,t) \) and \( \theta = \theta(x,t) \) are non-dimensional Cartesian velocity vector, pressure and temperature respectively. Equations 2.1-2.3 are the representation of the conservation of mass, momentum and energy respectively. The above equations are obtained by non-dimensionalizing the dimensional Navier-Stokes equations, in which velocity is non-dimonsialized by \( u_{ref} \), time by \( l_{ref}/u_{ref} \) and length by \( l^* = l_{ref} \).

The conventional choices for \( u_{ref} \) are, the jet average velocity, \( U_j \), and crossflow velocity, \( U_\infty \) and the conventional choices for \( l_{ref} \) are the jet diameter \( D \), the crossflow boundary
layer thickness $\delta$, momentum thickness $\delta_\theta$ and displacement thickness $\delta^*$ where $\delta$, $\delta_\theta$ and $\delta^*$ are dependent choices \textit{i.e.} there is a one-to-one relationship between $\delta$, $\delta_\theta$ and $\delta^*$. Overall according to Buckingham’s pi theorem four independent non-dimensional parameters exist for the current problem: (1) $Re = \frac{u_{ref}l_{ref}}{\nu}$ where $\nu$ is the kinematic viscosity; (2) blowing ratio $BR = \frac{U_j}{U_\infty}$; (3) the ratio of boundary layer (or momentum or displacement) thickness to jet diameter: $\delta/D$; (4) Prandtl number $Pr = \frac{\nu}{\alpha}$ where $\alpha$ is the thermal diffusivity. Temperature ($T$) is non-dimensionalized using

$$\theta(x,t) = \frac{T(x,t) - T_c}{T_h - T_c},$$

where $T_h$ is the crossflow temperature and $T_c$ is the jet temperature.

2.2 Spectral/hp element method

To solve equations 2.1-2.3, we perform Direct Numerical Simulation (DNS) in which all length scales and time scales are resolved without any use of turbulent models. For discretization in space, we use spectral/hp element method implemented in $N\epsilon\kappa\tau\alpha r$ [89]. The spectral/hp method has the geometric flexibility of h-type finite element method ($h$-convergence), while it benefits from the fast convergence of spectral methods ($p$-convergence) for a sufficiently smooth solution. These two different convergence strategies of spectral/hp element method make it a particularly attractive method for the film cooling problem studied in the current research. The geometry of the problem considered here includes a 35°-degree inclined pipe attached to a plenum. The sharp angles present at the edges where the inclined pipe is attached to the plenum or to the main domain result in very large amount of vorticity, thus necessitating an h-refinement strategy in these regions where the size of elements are reduced with order of polynomial fixed. Furthermore the fast convergence of spectral polynomials significantly reduces the size of the discrete problem for a given desired accuracy when compared to finite difference/volume methods.

In this section the components of the spectral/hp element method are reviewed. For a more detailed explanation see [50].
2.2.1 High-order splitting scheme

Equations 2.1-2.3 are advanced in time using a stiffly stable third order splitting scheme [49]. The semi-discrete system is given by:

\[
\frac{\hat{\mathbf{u}} - \sum_{q=0}^{J} \alpha_q \mathbf{u}^{n-q}}{\Delta t} = -\sum_{q=0}^{J} \beta_q [\mathbf{u} \cdot \nabla] \mathbf{u}^{n-q},
\]

(2.4)

\[
\frac{\hat{\mathbf{u}} - \mathbf{u}}{\Delta t} = -\nabla p^{n+1},
\]

(2.5)

\[
\frac{\gamma_0 \mathbf{u}^{n+1} - \hat{\mathbf{u}}}{\Delta t} = \frac{1}{Re} \nabla^2 \mathbf{u}^{n+1}.
\]

(2.6)

The coefficients \( \alpha_q, \beta_q \) and \( \gamma_0 \) are given in table 2.1. In equation 3.2, convective terms are integrated explicitly to obtain the intermediate velocity \( \hat{\mathbf{u}} \). In equation 2.5, pressure is obtained by enforcing that \( \hat{\mathbf{u}} \) is divergence-free which results in the Poisson equation for pressure as follows:

\[
\nabla^2 p^{n+1} = \nabla \cdot \left( \frac{\hat{\mathbf{u}}}{\Delta t} \right).
\]

(2.7)

The pressure-Poisson equation 2.7 is solved with the consistent boundary condition [50]:

\[
\frac{\partial p^{n+1}}{\partial n} = -\mathbf{n} \cdot \left[ \sum_{q=0}^{J-1} \beta_q [\mathbf{u} \cdot \nabla] \mathbf{u}^{n-q} + \frac{1}{Re} \sum_{q=0}^{J-1} \beta_q \nabla \times \omega^{n-q} \right].
\]

(2.8)

Finally diffusive terms are implicitly integrated as shown in equation 3.2 to obtain the updated velocity \( \mathbf{u}^{n+1} \).

The advection-diffusion equation is integrated in time similar to any component of the velocity field by only skipping equation 2.5.

2.2.2 Modal polynomial expansion

For space discretization modal polynomials within each element are used as the basis. We use hexahedral elements throughout this research. The three-dimensional basis are
Table 2.1: Weights for the stiffly-stable time integrator scheme (see [50], chapter 4.)

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>1st order ((J = 1))</th>
<th>2nd order ((J = 2))</th>
<th>3rd order ((J = 3))</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\gamma_0)</td>
<td>1</td>
<td>3/2</td>
<td>11/6</td>
</tr>
<tr>
<td>(\alpha_0)</td>
<td>1</td>
<td>2</td>
<td>3</td>
</tr>
<tr>
<td>(\alpha_1)</td>
<td>0</td>
<td>-1/2</td>
<td>-3/2</td>
</tr>
<tr>
<td>(\alpha_2)</td>
<td>0</td>
<td>0</td>
<td>1/3</td>
</tr>
<tr>
<td>(\beta_0)</td>
<td>1</td>
<td>2</td>
<td>3</td>
</tr>
<tr>
<td>(\beta_1)</td>
<td>0</td>
<td>-1</td>
<td>-3</td>
</tr>
<tr>
<td>(\beta_2)</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

obtained by the tensor product of the following one-dimensional basis [50]:

\[
\phi_p(\xi) = \begin{cases} 
\frac{1 - \xi}{2}, & p = 0, \\
\frac{-1 - \xi}{2}\left(\frac{1 + \xi}{2}\right)P_{p-1}^{1,1}, & 0 < p < P, \\
\frac{1 + \xi}{2}, & p = P,
\end{cases}
\]  

\[(2.9)\]

where \(P^{(\alpha,\beta)}\) are the Jacobi polynomials. The family of Jacobi polynomials are the solutions to a singular Sturm-Liouville problem in the region of \(-1 < x < 1\):

\[
\frac{d}{dx}\left[(1-x)^{1+\alpha}(1+x)^{1+\beta}\frac{d}{dx}P^{(\alpha,\beta)}_p(x)\right] = \lambda_p(1-x)^\alpha(1+x)^\beta P^{(\alpha,\beta)}_p(x),
\]

\[(2.10)\]

where \(\lambda_p = -p(\alpha + \beta + p + 1)\). An important characteristic of the Jacobi polynomials is their orthogonal relationship with respect to the weight \(w(x)\):

\[
(P^{(\alpha,\beta)}_p(x), P^{(\alpha,\beta)}_q(x))_w = C\delta_{pq}
\]

\[(2.11)\]

where \(\delta_{pq}\) is Kronecker delta and \(w(x)\) serves as the weight of the inner product in the following sense:

\[
(f, g)_w := \int_{-1}^{1} f(x)g(x)w(x)dx, \quad w(x) = (1-x)^\alpha(1+x)^\beta
\]
The one-dimensional basis 2.9 are shown in figure 2.1 for $P = 4$. Note that $\phi_0(\xi)$ and $\phi_P(\xi)$ are the only nonzero modes at the boundaries. These modes are referred to as **boundary modes**. The rest of the modes $\phi_p(\xi)$ for $0 < p < P$, that are zero at element boundaries, are **interior modes**. This setting ensures the continuity of the basis across elements ( $C^0$ – continuity).

The expansion to three dimension is obtained by tensor product of one-dimensional basis:

$$ u(\xi_1, \xi_2, \xi_3) = \sum_{p=0}^{P} \sum_{q=0}^{P} \sum_{r=0}^{P} \hat{u}_{pqr} \phi_p(\xi_1) \phi_q(\xi_2) \phi_r(\xi_3). $$(2.12)

### 2.3 Validation

The spectral element solver has been validated against numerous existing experimental and numerical cases and the validations have been documented. See [50] and references
therein for these cases. Here we consider two problem setting that are relevant to the
research undertaken in this dissertation.

2.3.1 Vertical jet in a crossflow

In the first example we solve a vertical jet in a crossflow to qualitatively validate the
solver for well-documented vortical structures in this problem. The schematic of the domain
and the boundary conditions used to solve equations 2.1-2.3 are shown in figure 2.2. The
origin of the coordinate system is at the center of the jet exit hole. The geometry includes
a vertical nozzle with the bottom diameter of 8D and top diameter of D, resulting in area
ratio of 64:1. The main domain, where the jet and free-stream interact, spans the volume of
23D×12D×6D in streamwise, wall-normal and spanwise directions respectively. The center
of the jet exit is located 8D downstream of the inlet boundary of the domain. For inflow
boundary condition, a laminar boundary layer profile with the boundary layer thickness of
δ99%/D = 2 and temperature of θ = 1 are used. At the top boundary, free-stream condition
is used with u = (U∞, 0, 0) and θ = 1. At the outflow boundary a zero-gradient condition is
assumed for both velocity and temperature. In the spanwise direction a periodic boundary
condition is enforced. For all of the wall boundaries, a no-slip boundary condition for
velocity and adiabatic boundary condition for temperature is used. At the nozzle inflow,
a spatially-uniform vertical velocity of u2 = αUj is specified, where α accounts for the
area ratio of the bottom of the nozzle (Ap) to the area of the jet exit. Since the diameter
ratio is 8:1, α = 1/64. The jet Reynolds number, Rej = UjD/ν is 2000 and the velocity
ratio R = 2, thus the crossflow Reynolds number is Re∞ = U∞D/ν = 1000. The Prandtl
number is Pr = 1.0.

A three-dimensional view of the computational grid that is used in this study is shown
in figure 2.3. The final grid obtained had 177452 elements. We use polynomial with spectral
order of four which translates into (4 + 1)³ = 125 points within each element, resulting in
approximately a total of 22 millions degrees of freedom.

In figure 2.4, instantaneous temperature at the mid-plane (x3 = 0) is shown. The shear
layer vortices are clearly visible at the jet upper shear layer. The upright vortices are can be seen in the jet wake. Other classical vortical structures such as horseshoe vortex, wall vortices can be clearly seen in figure 1.2. See chapter 1 for a more detailed explanation of these vortices.

### 2.3.2 Inclined jet in a crossflow

For inclined jet in crossflow we compare the results obtained from our direct numerical simulation with the experimental measurements carried out by Bidan et al. [13] for a 35-
degree inclined jet in crossflow. The computational domain consists of a pipe with length of 7$D$ with a 35-degree angle with the crossflow and a mainstream domain that expands the volume of 21$D$ in the streamwise direction, 10$D$ in the wall-normal direction and 6$D$ in the spanwise direction. The Reynolds number based on the crossflow velocity and the boundary layer thickness is $Re_\infty = U_\infty \delta/\nu = 1710$. The crossflow has a laminar boundary layer profile with a boundary layer thickness of 0.63$D$. At the pipe inlet uniform flow is superposed with 10% turbulence, generated by employing the spectral synthesizer method introduced in [85]. A hexahedral grid with 121,700 elements with a spectral polynomial degree of
Figure 2.4: Instantaneous temperature at mid-plane \((x_3 = 0)\) at velocity ratio \(R = 2\).

Figure 2.5: Comparison of time-averaged \(u_1\) velocity in the mid-plane with experimental data [13] at \(BR = 0.15\).

4 is used. The comparisons are made for a continuous case with \(BR = 0.15\). In figure 2.5 time-averaged streamwise velocity profiles are compared with those of experimental measurements in [13] at three different streamwise sections on the centerline plane. Overall good agreement is obtained and this gives us confidence in carrying out the computations with the current approach.
Chapter 3
The Effect of Plenum on Jet in Crossflow

The objective of the current chapter is to explore the effect of a supply plenum on the flow characteristics inside the delivery tube and its impact on near-field and far-field flow. We perform two direct numerical simulations for low velocity of $R = 0.4$ and $0.6$.

3.1 Problem statement

We solve the incompressible Navier-Stokes equations without any use of a turbulence model by resolving all time and space scales. These equations in non-dimensional form are:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u},$$

$$\nabla \cdot \mathbf{u} = 0,$$

where $\mathbf{u} = \mathbf{u}(\mathbf{x}, t) = \{u_1, u_2, u_3\}$ is the non-dimensional velocity vector field and $p = p(\mathbf{x}, t)$ is the non-dimensional pressure field in a space $(\mathbf{x} = \{x_1, x_2, x_3\})$ -time $(t)$. The Reynolds number is defined as $Re = U_\infty D/\nu$, where $U_\infty$ is the crossflow velocity, $D$ is the jet diameter and $\nu$ is the kinematic viscosity.

We also solve the advection-diffusion equation for passive scalar which is given by:

$$\frac{\partial S}{\partial t} + (\mathbf{u} \cdot \nabla) S = \frac{1}{ReSc} \nabla^2 S,$$

where $S$ is the scalar field and $Sc$ is the Schmidt number. In the current study we assume $Re = 1500$ and $Sc = 1$ for all cases.

The schematic of the problem is shown in figure 3.1. The origin of the coordinate system is at the center of the jet exit and $x_1$ is aligned with the streamwise direction, $x_2$ with the direction normal to the surface and $x_3$ with the spanwise direction. The plenum has the length, height and width of $7.9D$, $4D$ and $3D$ respectively. The length of the delivery tube
is 3.5D and the inclination angle is 35 degree. The mainstream domain, where the crossflow and the jet interact, spans the volume with the size of $18D \times 12D \times 3D$. The center of the jet exit is located $6D$ downstream from the crossflow inlet. The crossflow boundary condition is assumed to be a laminar boundary layer flow with boundary layer thickness of $\delta_{99\%}/D = 0.5$ where $\delta_{99\%}$ is the thickness that the streamwise velocity in the boundary layer reaches 99% of the free-stream velocity. On the top boundary plane, free-stream flow is assumed with $\mathbf{u}(x_1, 12, x_3) = (U_\infty, 0, 0)$ and $S(x_1, 12, x_3) = 1$, and on the spanwise direction periodic boundary condition is enforced. A zero normal derivative boundary condition is assumed at the outflow. The uniform boundary condition is used for the vertical velocity component $u_2$ at the bottom of the plenum with $u_1 = u_3 = 0$ and $S = 0$. The vertical velocity at the bottom of the plenum is equal to $u_2 = \alpha R$. Here $\alpha = \pi D^2/4A_p$, with $A_p$ being the area of the bottom of the plenum, accounts for the ratio of the area of the delivery tube to that of the plenum. The area scaling factor is $\alpha = 0.0331$.

3.2 Numerical method

In all the cases in this study we perform direct numerical simulation on the jet in cross-flow with the schematic of the problem shown in figure 3.1. The computational time step was roughly 0.001. All simulations were advanced at least $60D/U_\infty$ before the statistics are collected. The statistics are then collected for approximately another $60D/U_\infty$. Additional sampling did not change the result.

The details of the discretization in space and time are described in section 2.2. The semi-discrete equations given by equations - are advanced in time using the third-order splitting scheme explained in section 2.2.

3.2.1 Computational mesh

We use hexahedral elements of spectral order of four in the current simulations. Figure 5.4(a) shows the three-dimensional view of the grid. To generate the grid, quadrilateral meshes were first generated in the $x_1 - x_3$ planes in the main-stream domain, the delivery tube and the plenum. The quadrilateral elements exactly match across the common faces.
Figure 3.1: Schematic of the jet in the crossflow: (a) $x_1 - x_2$ view; (b) $x_1 - x_3$ view.

In between the volume parts. The two-dimensional mesh was then swept along the $x_2$ direction in the mainstream domain and plenum and along the axis of the cylinder in the delivery tube. The height of the first element in the crossflow boundary layer is $0.07D$. The height of the elements increases (with 16 elements) using hyperbolic distribution to $0.25D$ at $x_2 = 3$. Two elements were used from $x_2 = 3$ to the top boundary at $x_2 = 12$. Note that flow in this region remains steady with small gradients which justifies using large elements. Along the delivery tube, 35 elements with hyperbolic distribution were used. The height of the first and last element are $0.08D$. The finest elements are found in the delivery tube, especially in the boundary layer region where flow is highly unsteady with steep gradients. A close $x_1 - x_3$ view of the mesh in the vicinity of the jet exit is shown in figure 5.4(b).
3.3 Results

3.3.1 Flow inside the delivery tube

At both velocity ratios, flow has very small magnitudes of velocity almost everywhere inside the plenum due to the large cross-sectional area of the plenum compared to the pipe cross section, i.e. $A_p/\pi D^2/4 \simeq 33$. Only near the entrance of the delivery tube, flow accelerates as a result of mass conservation. The inclination of the delivery tube, results in an asymmetric flow condition at the horizontal plane at the inlet of the tube. In figure 3.3(a) the projected streamlines along with the velocity vectors at the inlet of the delivery tube at velocity ratio $R = 0.4$ are shown. Higher velocity magnitudes are observed near the downstream edge of the hole which is caused by the higher amount of “compression” of the
stream-tubes due to sharper change of angle compared to upstream edge of the hole. The sharp turn results in the separation of the flow at the downstream side of the delivery tube. A stream surface forms inside the delivery tube as shown schematically in figure 3.3(b) that splits the flow inside the delivery tube into two regions: (1) the upstream region with high velocity magnitudes as denoted by I in figure 3.3(b); and (2) the downstream region with low velocity magnitudes where an axial counter-rotating vortex pair (CVP) forms. In figure 3.3(c) the iso-surface of $\lambda_2 = -0.5$ (see [47] for the definition of $\lambda_2$) is shown with gray scale from black (negative axial vorticity) to white (positive axial vorticity). The CVP inside the tube is resulted by the inrush of flow from region I to region II, that passes from the small gap between the stream surface and the tube side walls. As a result of the compression of the projected streamlines, highest in-plane velocity magnitudes are observed at the two side gaps. The enclosed flow in region II forms a pair of counter-rotating vortices which entrains fluid from region I to region II as flow travels along the tube. The entrained mass results in increase in vertical velocity at higher vertical sections of the delivery tube.

In figure 3.4, the vertical velocity profiles at four horizontal sections along the tube are shown. All velocity profiles are taken at the mid-plane ($x_3 = 0$). The increase of vertical velocity in region II and its decrease in region I are shown by two red arrows. At the jet exit ($x_2 = 0$), the vertical velocity in region I still has higher velocity magnitudes compared to region II.

The stream surface shown in figure 3.3(b) also serves as a shear layer due to high velocity gradient across this layer as can be seen in velocity profiles shown in figure 3.4. The shear layer is clearly visible in figure 3.5, in which the volume rendering of the magnitude of velocity is shown. The shear layer gets closer to the upstream wall due to the mutual induction of CVP inside the tube that tends to move the CVP away from the downstream wall and closer to the upstream side.

Closer to the jet exit, the upper shear layer is pushed to the downstream side of the jet exit due to the high-pressure region (“pressure cap”) caused by the blockage of the
Figure 3.3: Flow conditions inside the delivery tube for velocity ratio $R = 0.4$: (a) projected streamlines and velocity vectors on the inlet of the delivery tube at $x_2 = -2$; (b) authors’ representation of flow inside the delivery tube; (c) velocity profile at mid-section $x_2 = -1$. The cross section is perpendicular to the pipe axis. Also the iso-surface of $\lambda_2 = -0.5$ is shown whose color is in gray scale with black (negative axial vorticity) to white (positive axial vorticity).

crossflow. As a result of mass conservation and partial blockage of the flow at the upper shear layer, the flow accelerates in region II near the jet exit.

At velocity ratio $R = 0.6$ the CVP inside the delivery tube becomes stronger and become unstable inside the delivery tube. The oscillations in the shear layer inside the delivery tube are shown in figure 3.6. In figure 3.7 the $\lambda_2$-isosurface for velocity ratio $R = 0.6$ are shown. The oscillations inside the delivery tube form clear and distinct hairpin vortices as it can be seen in figure 3.7.
3.3.2 Near and far field flow

Figure 3.8 shows the volume rendering of the scalar field at velocity ratio $R = 0.4$. The CVP resulting from the spillage of the jet fluid from the side walls of the tube can be
oscillations in shear layer

Figure 3.6: Volume rendering of velocity magnitudes at velocity ratio $R = 0.6$ from $x_1 - x_2$ view.

hairpin vortices

Figure 3.7: Close up view of $\lambda_2$ iso-surfaces in the delivery tube at $R = 0.6$ for inclined jet in crossflow with a plenum. The gray scale is used black (negative streamwise vorticity) to white (positive streamwise vorticity).
Figure 3.8: Volume rendering of scalar field at velocity ratio $R = 0.4$ for inclined jet in crossflow with a plenum.

clearly seen in figure 3.8(a). Due to mutual induction of the CVP, the distance between two counter-rotating vortices decreases as the flow convects downstream. The closer the CVP gets to each other, the stronger the vertical velocity, that they induce on each other, becomes. As a result, the CVP rises as it also stretches in the wall boundary layer of the background flow. The stretching results in faster convection of the upper part of the CVP compared to the lower part. The CVP becomes unstable and oscillations appear caused by instability are shown in figure 3.8(b). These observations are in agreement with the experimental measurements carried out by Acarlar and Smith [2]. The oscillations grow and form hairpin vortices which are responsible for convecting the jet fluid further downstream. The hairpin vortices are the dominant vortical structures at low velocity ratios, which is in contrast with high-velocity-ratio JICFs. At higher velocity ratios, the
vortical structures are fundamentally different as discussed in chapter 1.

At velocity ratio $R = 0.6$, the hairpin vortices are formed inside the delivery tube, reach to the upper and lower shear layers. Figure 3.9 shows the volume rendering of the scalar field at velocity ratio $R = 0.6$. The vortex rollup near the jet exit ($x_1 < 2$) is due to the excitation caused by hairpin vortices traveling along the delivery tube and reach the shear layer. These vortices turn into hairpin vortices approximately at $x_1 \simeq 4$ as shown in figure 3.9(b). At velocity ratio $R = 0.6$, the CVP is stronger than that of $R = 0.4$. The CVP are shown in figure 3.9(a).
Chapter 4
Optimization of forcing parameters of film cooling effectiveness

Based on the results reported in chapter 1 and the arguments therein, it is clear that the role of pulsation on the cooling effectiveness needs to be better understood. In the current study, we perform controlled numerical experiments using high-fidelity direct numerical simulations for a wide range of pulsation frequencies ($St \in [0.2, 2]$) and duty cycles ($DC \in [0.05, 1]$). We extract the response surface of the system with respect to $DC$ and $St$, and identify global and local optimums in the $DC – St$ space. In all of our simulations plenum and the delivery tube are included, since these are recognized to play an improvement role on the film cooling behavior [17, 58, 14]. To overcome the expensive computational cost of such a problem, we employ an efficient discretization strategy in the design space by using fast-converging spectral polynomials along with a high-order spectral element method to carry out the direct numerical simulations. Within the range of parameters considered, a key goal of the paper is to find the optimal pulsation parameters, and to understand the mechanisms leading to the global optimal condition for film cooling effectiveness. A description of this optimization approach, and the corresponding results obtained for the film cooling cases are presented in this chapter.

4.1 Methodology

The incompressible Navier-Stokes/energy equations as given by 2.1-2.3 are solved without any use of a turbulence model by resolving all time and space scales.

We consider $u = u(x, t; \xi)$, $p = p(x, t; \xi)$ and $\theta = \theta(x, t; \xi)$ where $\xi = (\xi_1, \xi_2)$ is a two-dimensional vector of design or control parameters. In the current study, design parameters are the duty cycle $DC$, and the non-dimensional pulsation period $T_p = U_\infty / D f$, where $U_\infty$ is the crossflow velocity, $D$ is the jet diameter and $f$ is the dimensional pulsation frequency.
4D
3D
7.9D

free stream
outflow
periodic
blade surface

\[ u_1 = f(x_2) \]
\[ \theta = 1 \]

plenum inflow
\[ u_2 = g(t; \xi) \]
\[ u_1 = u_3 = \theta = 0 \]

As it will be explained later in this section \( DC \) and \( T_p \) are related to \( \xi_1 \) and \( \xi_2 \) through a linear mapping. The Reynolds number is defined as \( Re = U_\infty D/\nu \), where \( \nu \) is the kinematic viscosity. Figure 5.1 shows the schematic of the problem. The velocity signal at the bottom of the plenum, \( g(t; \xi) \) is fixed by specifying the design parameters \((DC, T_p)\) and the maximum and minimum plenum-inflow velocity. In figure 4.2, the generic form of the signal \( g(t; \xi) \) is shown. The amount of the time that the jet is on within each full period is denoted by \( \Delta t_{on} \) and therefore \( DC = \Delta t_{on}/T_p \). We also introduce \( \tau \) that shows the time percentage during a cycle. Therefore \( \tau < DC \) represents the period that the jet is on and

Figure 4.1: Schematic of the jet in crossflow.
\[ \tau > DC \] when the jet is off. When the jet is on, the vertical velocity at the bottom of the plenum is equal to \( \alpha \)BR. Here \( \alpha = \pi D^2/4A_p \), with \( A_p \) being the area of the bottom of the plenum, accounts for the ratio of the area of the delivery tube to that of the plenum.

When the jet is off, it is turned completely off and during this time the net mass flow rate through any \( x_2 \)-section at the plenum or delivery tube is zero.

### 4.1.1 Optimization problem

Our objective is to maximize the film cooling effectiveness as a function of duty cycle and pulsation period. To measure the effectiveness of film cooling we first define the spanwise-averaged film cooling effectiveness which is given by:

\[
\eta(x_1; \xi) = \frac{1}{w(t_f - t_i)} \int_{t_i}^{t_f} \int_{-w/2}^{w/2} (1 - \theta(x_1, t; \xi)) \mid_{x_2=0} \, dx_3 \, dt, \tag{4.1}
\]

with \( w \) being the width of the cooled surface, \( t_i \) and \( t_f \) are the beginning and the end of the period over which the time-averaged quantities are calculated. In order to obtain an overall measure of the film cooling effectiveness, we further calculate the streamwise
average of $\eta(x_1; \xi)$. This results:

$$\bar{\eta}(\xi) = \frac{1}{x_{1e} - x_{1s}} \int_{x_{1s}}^{x_{1e}} \eta(x_1; \xi) dx_1,$$  \hspace{1cm} (4.2)$$

where $x_{1s}$ and $x_{1e}$ are the two ends of the interval in the streamwise direction within which the average film cooling effectiveness is calculated. The overall film cooling effectiveness, $\bar{\eta}(\xi)$, is a function of two independent variables $DC$ and $T_p$, with $BR$ and geometry fixed. The goal is to find the $DC$ and $T_p$ at which the average film cooling effectiveness, $\bar{\eta}(\xi)$, is globally maximum, and to understand the reasons that lead to this condition.

### 4.1.2 Discretization in space-time

The details of discretization in space and time are given in section 2.2.

### 4.1.3 Discretization in design parameter space

We intend to estimate the response surface for film cooling pulsation where the design parameters, $DC$ and $T_p$, constitute a Cartesian two-dimensional space and the response surface is $f := f(x; \xi)$. Here $f$ is a generic time-averaged variable. In order to discretize the governing equations 3.1-5.3 in the design space $(\xi_1, \xi_2)$, we use pseudo-spectral method [92] to project $f(x; \xi)$ to a polynomial space. This follows:

$$f(x; \xi) = f_N(x; \xi) + \epsilon(x; \xi),$$  \hspace{1cm} (4.3)$$

with

$$f_N(x; \xi) = \sum_{j=0}^{P_2} \sum_{i=0}^{P_1} \hat{f}_{ij}(x) \phi_i(\xi_1) \phi_j(\xi_2),$$  \hspace{1cm} (4.4)$$

where $\phi_i(\xi_1)$ and $\phi_j(\xi_2)$ are polynomials up to degree $P_1$ and $P_2$ respectively, and $\epsilon(x; \xi)$ is the error of the expansion. For convenience we use a two-dimensional basis which is obtained by multiplying the univariate basis in each direction. This follows:

$$\phi_r(\xi) = \phi_i(\xi_1) \phi_j(\xi_2), \hspace{0.5cm} 0 \leq i \leq P_1, \hspace{0.5cm} 0 \leq j \leq P_2,$$  \hspace{1cm} (4.5)$$
where \( r \) is:

\[
r = i + j(P_1 + 1), \quad 0 \leq i \leq P_1, \quad 0 \leq j \leq P_2.
\]

Therefore \( 0 \leq r \leq N \) with \( N = (P_1 + 1)(P_2 + 1) \). Thus equation 4.4 can be re-written as:

\[
f_N(x; \xi) = \sum_{i=0}^{N} \hat{f}_i(x) \phi_i(\xi). \tag{4.6}
\]

We use Legendre polynomials for their fast convergence for a smooth function \( f \). We define an inner product and a norm given by:

\[
(f, g) := \int f(\xi) g(\xi) d\xi, \quad \|f\| := (f, f)^{1/2}. \tag{4.7}
\]

Legendre polynomials form an orthogonal basis, and therefore:

\[
(\phi_i, \phi_j) = \delta_{ij} \gamma^2_j, \quad i, j = 0, 1, 2, \ldots, N, \tag{4.8}
\]

where \( \gamma_j = \|\phi_j\| \).

The coefficients \( \hat{f}_i(x) \) are obtained using Galerkin projection by taking the inner product of two sides of equation 4.6 with \( \phi_j \). Using the orthogonality relation of the Legendre polynomial given by equation 4.8, we have:

\[
\hat{f}_j(x) = \frac{\int f(x; \xi) \phi_j(\xi) d\xi}{\gamma^2_j}, \quad j = 0, 1, 2, \ldots, N. \tag{4.9}
\]

To calculate \( \int f(x; \xi) d\xi \), we use pseudo-spectral method [92] in which the integral is estimated using a quadrature rule. This follows:

\[
\int f(x; \xi) \phi_j(\xi) d\xi \simeq \sum_{i=1}^{Q} w^{(i)} f(\xi^{(i)}) \phi_j(\xi^{(i)}), \tag{4.10}
\]
where $\xi^{(i)}$ and $w^{(i)}$, $i = 1, 2, \ldots Q$ are quadrature points and weights in the two-dimensional space of design variables respectively.

We use Clenshaw-Curtis nodes [19] which are the extrema of Chebychev polynomials and for a one-dimensional Chebyshev polynomial of order $P$ are given by:

$$\Theta^P = \{\cos\frac{2\pi i}{P}, i = 0, 1, \ldots, P\}. \quad (4.11)$$

Using Clenshaw-Curtis nodes ($\Theta^P$) results in fast convergence of $f_N(x; \xi)$ to $f(x; \xi)$ for a smooth function $f$ [10]. This fact is at the core of the efficiency of the method that we are using to discretize equations 3.1-5.3 in the design space. Since evaluating at each sample point is in fact solving an expensive DNS simulation, our goal is to minimize the number of samples for a desirable accuracy.

Following the above notation, if the Clenshaw-Curtis nodes for a polynomial of order $P$ in the $\xi_r$-direction are denoted by $\Theta^P_r$, the Clenshaw-Curtis nodes in the two-dimensional design space are given by the full tensor product of the corresponding one-dimensional nodes in each direction. This follows:

$$\Theta^{(P_1, P_2)} = \Theta^{P_1}_1 \otimes \Theta^{P_2}_2. \quad (4.12)$$

In this study we always use $P_1 = P_2 = P$, however, in general, they can be different. This results in $Q = (P + 1)^2$ quadrature points in the design space. Note that

$$\xi_i \in [-1, 1], \quad i = 1, 2.$$ 

A linear mapping is then used to transform $\xi_i$'s to arbitrary intervals. This follows:

$$\xi^*_i = \frac{1 - \xi_i}{2} a_i + \frac{1 + \xi_i}{2} b_i, \quad i = 1, 2,$$
where

$$\xi_i^* \in [a_i, b_i], \quad i = 1, 2.$$  

In the current study $\xi_1^*$ and $\xi_2^*$ represent pulsation period ($T_p$) and duty cycle (DC) respectively. Now we can use a similar polynomial expansion as in the equation 4.6, to estimate the time-averaged quantities:

$$u_N(x; \xi) = \sum_{k=0}^{N} \hat{u}_k(x) \phi_k(\xi),$$  

(4.13)

$$p_N(x; \xi) = \sum_{k=0}^{N} \hat{p}_k(x) \phi_k(\xi),$$  

(4.14)

$$\theta_N(x; \xi) = \sum_{k=0}^{N} \hat{\theta}_k(x) \phi_k(\xi),$$  

(4.15)

where the expansion coefficients are obtained from:

$$\hat{u}_k(x) = \sum_{i=1}^{Q} w^{(i)}(x; \xi^{(i)}) \phi_k(\xi^{(i)})/\gamma_k^2, \quad 0 \leq k \leq N,$$  

(4.16)

$$\hat{p}_k(x) = \sum_{i=1}^{Q} w^{(i)}(x; \xi^{(i)}) \phi_k(\xi^{(i)})/\gamma_k^2, \quad 0 \leq k \leq N,$$  

(4.17)

$$\hat{\theta}_k(x) = \sum_{i=1}^{Q} w^{(i)}(x; \xi^{(i)}) \phi_k(\xi^{(i)})/\gamma_k^2, \quad 0 \leq k \leq N.$$  

(4.18)

Similarly, $\tilde{\eta}(\xi)$ is approximated using the polynomial expansion. This follows:

$$\tilde{\eta}_N(\xi) = \sum_{k=0}^{N} \tilde{\eta}_k \phi_k(\xi).$$  

(4.19)

Using equations 5.5, 5.6 and 4.19, it is easy to verify that:

$$\tilde{\eta}_k = 1 - \frac{1}{w(x_1 - x_{1s})} \int_{-w/2}^{w/2} \int_{x_{1s}}^{x_{1e}} \tilde{\theta}_k(x) | \theta_{x_2=0} dx_1 dx_3.$$  

(4.20)
4.2 Simulation details

In all the cases in this study we perform direct numerical simulation on the jet in crossflow with the schematic of the problem shown in figure 5.1. The geometry and boundary conditions of the problem are equivalent to those of presented in section 3.1. Only differences are mentioned here.

The mainstream domain spans the volume with the size of $22D \times 10D \times 3D$ with center of the jet exit is located $7D$ downstream from the domain inlet. The uniform boundary condition $g(t; \xi)$ (see figure 4.2) is used for the vertical velocity component $u_2$ at the bottom of the plenum with $u_1 = u_3 = 0$ and $\theta = 0$. As noted earlier, each pair of the design variables $(DC, Tp)$, and the peak blowing ratio (the blowing ratio when the jet in on and is set to 1.5) specifies the signal $g(t; \xi)$. The Reynolds number, $Re_j = U_j D/\nu$, based on the jet space-averaged velocity (when the jet is on) is 1500 throughout and Prandtl number is $Pr = 1$. All simulations were advanced at least $40D/U_\infty$ before the statistics are collected. The statistics are then collected for approximately another $40D/U_\infty$. Care was taken to ensure that the calculation of the statistics starts at the beginning of a pulsation cycle and finishes at the end of another pulsation cycle.

4.2.1 Computational mesh

We use hexahedral elements of spectral order of three and four in the current simulations. Figure 5.4(a) shows the three-dimensional view of the grid and a closeup view near the jet exit is shown in figure 5.4(b). The grid was generated with same strategy as presented in section 3.2.1. In each element spectral polynomials with order three or four were used. This amounts to a total degree of freedom of 3.1 millions for spectral order three, and 6.0 millions for spectral order of four.

In the design space, we use polynomial order up to degree $P=8$ for each design variable. This results in $(P + 1)^2 = 81$ points in the design space. In total, 73 DNS runs were performed, each on 56 processors and for 6 days. For all of the simulations studied in this work, roughly 600,000 CPU hours were consumed.
Figure 4.3: Unstructured hexahedral grid; (a) three-dimensional view; (b) $x_1 - x_3$ view of the grid in the vicinity of the jet exit with spectral order $m = 4$. 
Table 4.1: Grid convergence cases.

<table>
<thead>
<tr>
<th>Case</th>
<th>Case I</th>
<th>Case II</th>
<th>Case III</th>
<th>Case IV</th>
</tr>
</thead>
<tbody>
<tr>
<td>element size</td>
<td>48162</td>
<td>48162</td>
<td>48162</td>
<td>48162</td>
</tr>
<tr>
<td>spectral order</td>
<td>3</td>
<td>4</td>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td>(T_p)</td>
<td>2.75</td>
<td>2.75</td>
<td>steady</td>
<td>steady</td>
</tr>
<tr>
<td>DC</td>
<td>0.52</td>
<td>0.52</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

Figure 4.4: Grid convergence study for center-line film and spanwise averaged cooling effectiveness.

4.2.2 Convergence in space and time

To study the dependence of the numerical solution to grid resolution, we considered four simulations whose specifications are shown in table 7.2. As a refinement strategy, we increased the spectral order of the mesh from three to four. As explained in the previous section, the mesh with spectral order four has almost twice as many degrees of freedom as that of with the spectral order three. The centerline and spanwise-averaged film cooling effectiveness for all four simulations are shown in figure 4.4. The comparison are extended to also include the hole exit. The results show a consistent agreement in all cases considered. In the rest of this study, we use elements with spectral order three which provides proper
resolution for the current problem.

4.2.3 Convergence in design space

To study the convergence of the polynomial expansion for film cooling effectiveness, we project \( \tilde{\eta}(\xi) \) to the polynomial expansion given by equation 4.19 for increasing polynomial order, ranging from \( P = 0 \) to \( P = 8 \). Since we use equal polynomial order in \( \xi_1 \) and \( \xi_2 \) directions, therefore \( N = (P + 1)^2 \) and by varying \( P \), \( N \) changes accordingly. To measure the contribution by increasing polynomial order from \( P - 1 \) to \( P \), we calculate the modules \( m(P) \) as in the following:

\[
m(P) = \left( \sum_{k=0}^{(P+1)^2} \gamma_k^2 \tilde{\eta}_k^2 - \sum_{k=0}^{P^2} \gamma_k^2 \tilde{\eta}_k^2 \right)^{1/2} \quad 1 \leq P \leq 8, \quad (4.21)
\]

\[
m(P) = \gamma_0 |\tilde{\eta}_0| \quad P = 0.
\]

Here the \( \tilde{\eta}_k \)'s are the expansion coefficients in overall cooling effectiveness given by equation 4.20, and \( \gamma_k \) are used as a normalizing factor for the Legendre polynomials. The modules \( m(P) \) vs. \( P \) is shown in figure 4.5. The most significant contributions to the response surface are gained from polynomials in the range of \( P = 0 \) to \( P = 3 \). Larger polynomial contributions have modules of the same order, but almost one order smaller than those of lower polynomial orders. The main reason that the modules for the higher-order polynomials do not decay as fast as those of lower-order is the non-monotonic behavior of the average film cooling effectiveness vs. \( DC \) and \( T_p \). However the improvement of the response surface by increasing the polynomial order is marginal \((O(10^{-3}))\) and the behavior of the surface is essentially the same. Note that further increase in polynomial order requires prohibitively expensive DNS computations, while marginally modifying the response surface. In the rest of this study, we consider the response surface with polynomial order \( P = 8 \).
4.3 Results and discussions

4.3.1 Effect of pulsation on film cooling performance

In examining these results, it is important to remember that the blowing ratio fluctuates between 1.5, when the valve is open, and 0, when the valve is closed, and any comparisons with the steady $BR = 1.5$ case is therefore not at a constant coolant mass injection rate when integrated over a time duration. For the pulsed cases, the integrated coolant injection rates are lower, and an equal or greater cooling effectiveness than the baseline steady jet of $BR = 1.5$ represents an improvement in the cooling strategy. We choose the pulsation period to be in the range of $T_p \in [0.5, 5]$, which corresponds to Strouhal number in the range of $St \in [0.2, 2]$, and duty cycle in the range of $DC \in [0.05, 1]$. All the sampling points with $DC = 1$ correspond to a steady jet and therefore only one simulation was performed for these points.

Figure 5.7 shows the non-dimensional temperature at the $x_2$-plane which represents the cooling effectiveness on the surface and the flow temperature at the exit of the coolant hole. As noted earlier, the two control parameters being varied include the duty cycle...
Figure 4.6: Time-averaged temperature contours for quadrature points on cooled surface ($x_2 = 0$). Each row: constant $T_p$; Each column: constant $DC$. 
and the time period of pulsation \( T_p \) which is inversely proportional to frequency or Strouhal number. The rightmost column represents the steady blowing condition at a BR of 1.5. The signature of the near-hole jet blow off and the downstream spread of the coolant jet on the surface can be seen. Both duty cycle and time period have important and significant influence on the cooling effectiveness. In figure 5.7, it is observed that the cooling effectiveness generally improves at a lower duty cycle (which represents a lower amount of integrated coolant injection) and higher frequencies, and that optimal conditions exist with global maxima in the lower left quadrant of the \( DC - T_p \) design space within the range of design parameters considered. However, the integrated values of the cooling effectiveness do not show any distinct linear or monotonic behavior in the design space, and local maxima or peaks are obtained at other points as shown later in figure 4.13. More discussions on the mechanisms leading to the global maxima are discussed in the next section for the case with \( DC = 0.09 \) and \( T_p = 1.16 \), which is a point close to the global-maxima condition. In figure 4.7 the temperature distributions at \( T_p = 1.16 \) for increasing \( DC \) from 0.09 to 1 (steady injection) are shown. One can see that in the baseline condition (\( DC = 1 \)), the coolant jet is characterized by unsteady vortical structures that entrain and mix out the crossflow. The crossflow penetration to the surface is clearly evident in the longitudinal structures at elevated temperatures near the surface. As \( DC \) is reduced from 1, the turbulent structures and the near-wall thermal field are clearly influenced, with the lower duty cycles showing better organized structures and lower temperatures (presence of more coolant) near the wall. For \( DC \) larger than 0.34, the jet lift-off is observed immediately after entering the main domain resulting in poor coverage of the surface; this can also be seen in figure 5.7 where both the lateral and streamwise coverage of the coolant is poor. As \( DC \) is further decreased the coolant remains close to the surface and the crossflow entrainment under the coolant jet and close to the surface is reduced, and therefore lower coolant temperatures near the surfaces are observed. Thus the film cooling provides an effective coverage. These observations translate to higher cooling effectiveness as it is also observed in figure 5.7.
Figure 4.7: Instantaneous temperature surface in the mid-plane ($x_3 = 0$) with constant pulsation period of $T_p = 1.16$.

Further discussions on why the lower DC’s (where the integrated coolant mass flowrate is lower) leads to improved cooling effectiveness is provided later, and is related to the interaction between the mainstream and the coolant flow leaving the delivery tube. Figure 4.8 shows the instantaneous temperature distribution in the vertical mid-plane ($x_3 = 0$)
Figure 4.8: Instantaneous temperature surface in the mid-plane ($x_3 = 0$) with constant duty cycle of $DC = 0.52$.

at $DC = 0.52$ with increasing time periods from 0.67 to 5. At higher time periods of $T_p \geq 1.89$, the coolant jet exhibits discrete vortex rings that are formed during the on-protron of the pulsation cycle. The same vortex structures were also recognized by Sau and Mahesh [82] for a pulsed vertical jet. These vortex rings are responsible for carrying
the coolant flow downstream of the jet hole. The distance between successive vortex rings increases as the pulsation frequency decreases or the time period increases. Note that the instantaneous blowing ratio during the on-portion of the cycle is \( BR = 1.5 \) which roughly becomes the velocity at which the vortex rings are convected along the axis of the delivery tube (35 degree with \( x_1 \)). As a result, in figure 4.8, a distance of \( d = 1.5 T_p \) can be observed between two successive vortex rings. Since \( BR \) and \( DC \) remain unchanged for all the cases shown in figure 4.8, the higher pulsation period translates to more amount of the injected coolant, leading to the formation of larger and stronger vortex rings and greater instantaneous vertical-penetration. A visible trailing column can be seen for \( T_p \geq 3.61 \).

For the case with \( T_p = 4.34 \) the upper shear layer shows strong oscillations, caused by the traveling vortices created when the flow leaves the plenum and enters the delivery tube and undergoes a sharp turn. As a result an unsteady separation bubble forms in the upstream side of the tube, generating vortical structures that for a large enough pulsation period, such as \( T_p = 4.34 \), can travel the pipe length of 3.5\( D \) and affect the upper shear layer; a phenomena that is absent in smaller pulsation periods.

4.3.2 Optimal and sub-optimal behavior

In this section we investigate three cases in more details. In the previous section it was clear that lower duty cycle and high pulsation frequency (lower \( T_p \)) leads to higher film cooling effectiveness. The first case that we consider has \( DC = 0.09 \) and \( T_p = 1.16 \) which lies in the high film-cooling-effectiveness region in the design space and shows near-optimal film cooling performance (see figure 4.13). Temperature contours at four time instants of one pulsation cycle for this case are shown in figure 4.9. Note that \( \tau \) represents the percentage of time during each cycle with \( \tau = 0\% \) being the beginning and \( \tau = 100\% \) the end of the cycle, and thus for \( \tau < DC \) the jet is on and for \( \tau > DC \) the jet is completely off. At \( \tau = 4\% \) the jet is on and the injected coolant initiates hairpin vortices (shown by arrows in figure 4.9) in the shear layer. The hairpin vortices are periodically generated and convected downstream. As the valve closes at \( \tau = 9\% \), the crossflow enters the delivery
tube from the leading edge of the hole, pushing the coolant in the delivery tube towards the downstream side of the hole and the coolant leaves the tube from the downstream edge of the hole (this adjustment in the exit flow is driven by mass conservation since the plenum-inflow was abruptly shut off). The ingestion of the crossflow into the delivery tube continues by forming a vortex ring that is clearly visible in the tube and near the exit at \( \tau = 34\% \) and \( \tau = 64\% \). The ingested gas reaches very close to the opposite wall of the delivery tube at \( \tau = 94\% \). Note that the speed of the propagation of the vortex ring inside
the delivery tube is slightly less than the non-dimensional crossflow velocity of one (due to entrainment), and during the off-portion of the cycle with non-dimensional $\Delta t_{off} = 1.06$ (corresponding to $DC$ of 0.09), the vortex ring has long enough time to travel one non-dimensional unit of length (tube diameter) across the delivery tube and nearly reaches the opposite wall ($\tau = 94\%$). However, due to the short pulsation period, the plenum inflow is re-initiated before the ingested flow penetrates fully to the downstream-edge of the delivery tube, and before the ingested crossflow gas can cause a significant temperature rise in the interior wall of the delivery tube; this can be seen at the beginning of the new cycle at time $\tau = 4\%$ in figure 4.9.

Other than the hairpin vortices, the steady presence of a counter rotating vortex pair (CRVP) is observed. Figure 4.10 shows the lateral-vertical velocity vectors at two streamwise sections. A double-deck vortex pair is observed with the lower deck showing the CRVP and the upper deck is resulted by crossing the hairpin vortex legs. The velocity field that is induced by each pair of these vortex structures tend to lift the jet off the surface (see for example [41]) and unfavorably affects the cooling performance. The amount of the induced vertical velocity, however, is directly proportional to the strength of the vortex pair which is supplied by the amount of the coolant injection. Due to both short duty cycle ($DC = 0.09$) and pulsation period ($T_p = 1.16$), the induced vertical velocity for this case is small (0.01 of the crossflow velocity) resulting in slow rise of the coolant from the surface (see the first image in figure 4.7) and thus providing an effective coverage of the surface.

The second case we discuss here is the pulsed film cooling with higher duty cycle of $DC = 0.34$ compared to the first case and with the same pulsation period of $T_p = 1.16$. Four snapshots during one pulsation period are shown in figure 4.11. The top images show the injection of the coolant into the crossflow. The coolant forms a vortex ring that gains more strength as the time progresses during the on-portion of the cycle and the size of the vortex ring increases from $\tau = 10\%$ to $\tau = 27\%$. In this case larger duty cycle injects more mass flow rate during the on-period and allows the complete formation of a vortex
ring, an observation in contrast with the case at $DC = 0.09$ where the smaller duty cycle is not long enough to allow the formation of vortex rings during the injection time. The vortex rings, which carry significant amount of the coolant, penetrate the mainstream flow and are separated from the surface, thus allowing the entrainment of the crossflow to the surface below the jet. This leads to poor coverage of the surface as it can be seen in figure 5.7. During the off-portion of the cycle the initial stages of the crossflow ingestion into the delivery tube can be observed in figure 4.11 at $\tau = 57\%$ and $\tau = 87\%$ by the rolling up inside the hole. However, due to the low pulsation period ($T_p = 1.16$) and small non-dimensional time length in the off-portion of the cycle ($\Delta t_{off} = 0.77$), the hot gas cannot reach the tube walls before the next cycle begins. This shows that despite the ingestion of the hot gas into delivery tube, the wall temperature does not increase.

The third case investigated in this work is at $DC = 0.09$ and $T_p = 2.75$, which has the same duty cycle as that of the first case but higher pulsation time period. Four instants of temperature contours during one cycle are shown in figure 4.12. During the on-portion of the cycle ($\tau = 9\%$), a starting vortex forms which gains more strength compared to that of the first case (due to the longer $\Delta t_{on}$ for this case) and grows larger in size, entraining more amount the hot mainstream flow, leading to increased mixing. During the off-portion of the cycle, a vortex ring similar to the first case forms, causing the ingestion of the crossflow into
Figure 4.11: Snapshots of instantaneous temperature at $x_3 = 0$ during one pulsation cycle with $DC = 0.34$, $T_p = 1.16$, $\Delta t_{on} = 0.39$ and $\Delta t_{off} = 0.77$.

The longer amount of the pulsation period ($\Delta t_{off} = 2.50$) compared to the first case ($\Delta t_{off} = 1.06$) gives the ingested gas enough time to impinge to the tube wall, causing significant increase in the wall temperature. Between two successive pulses, the crossflow penetrates to the delivery tube surface due to larger lapses between the coolant pulses and thus adversely affecting the cooling coverage.

The last two cases demonstrate the mechanisms that are responsible for the deterioration of the film cooling effectiveness when either duty cycle or pulsation period increases
with respect to the near-optimal case ($DC = 0.09$ and $T_p = 1.16$). We also note that for pulsed cases with very small duty cycle ($DC < 0.05$), the film cooling coverage is severely affected by the overwhelming amount of the crossflow compared to the coolant. Also for lower pulsation periods than $T_p = 1.16$ the overall film cooling effectiveness slightly decreases, but the physical mechanisms in the flow remain similar to the first case. Thus clearly, optimal conditions exists for the pulsation parameters, and the mechanisms controlling this are related to the dynamics near the jet-hole exit.
4.3.3 Film cooling response surface

In figure 4.13, the contour lines of the space- and time-averaged film cooling effectiveness vs. $DC$ and $T_p$ are shown. The global optimum is shown by point P1 and its design parameters are at $DC = 0.14$ and $T_p = 1.16$, which are close to those of the first case discussed in the previous section including the mechanisms responsible for such a behavior. Local optimums are denoted by points P2 to P5. The coordinate of points P1-P5 are their respective values of averaged film cooling effectiveness are given in table 4.2.

From the computational point of view an important observation is the presence of local optimums. This non-convex behavior of the objective function versus the design variables renders the gradient-based optimization strategies inappropriate for this problem, since the gradient-based algorithms will converge to a local optimum depending on the initial guess. This can be partially remedied by using different initial guesses. However for the current problem each initial guess leads to several expensive DNS simulations. Nevertheless certainty in obtaining the global optimum cannot be guaranteed in such approaches. Moreover the direct numerical simulations required in gradient-based approaches, have to be carried out in sequence due to the inherent dependence of the updated guess to previous iterations. In contrast, in the current approach all sample points are independent of each other and all simulations can be carried out concurrently, resulting in a significant scale up in the total computation performance.

Another advantage of the current approach, although not exploited in the this study, lies in the fact that the obtained response surface can play the role of a surrogate model and can be utilized in different design scenarios, such as optimization cases where the objective function is the film cooling effectiveness with the amount of the coolant penalized with different factors, or the amount of ingestion penalized and so on. Note that once the response surface is obtained, investigating different optimization problems can be carried out very quickly without incurring the computational cost of the DNS.
4.4 Conclusions

In this chapter, we investigated the influence of duty cycle and pulsation frequency on film cooling effectiveness for a 35-degree inclined jet in crossflow with a plenum attached to the delivery pipe. We have presented an efficient computational strategy that combines high-fidelity simulations and construction of a response surface in the design space to find the optimal forcing parameters that maximizes the film cooling effectiveness. We carried out 73 direct numerical simulations and obtained the response surface of the averaged film cooling effectiveness as a function of the two design variables of duty cycle and pulsation period for a wide range of \( DC \in [0.05, 1] \) and \( T_p \in [0.5, 5] \). In summary the findings of this
Table 4.2: Optimal design points.

<table>
<thead>
<tr>
<th>Point</th>
<th>DC</th>
<th>$T_p$</th>
<th>St</th>
<th>$\bar{\eta}_N(\xi)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>P1</td>
<td>0.14</td>
<td>0.97</td>
<td>1.03</td>
<td>0.18</td>
</tr>
<tr>
<td>P2</td>
<td>0.38</td>
<td>2.43</td>
<td>0.41</td>
<td>0.08</td>
</tr>
<tr>
<td>P3</td>
<td>0.56</td>
<td>1.21</td>
<td>0.83</td>
<td>0.076</td>
</tr>
<tr>
<td>P4</td>
<td>0.24</td>
<td>4.54</td>
<td>0.22</td>
<td>0.075</td>
</tr>
<tr>
<td>P5</td>
<td>0.54</td>
<td>4.84</td>
<td>0.21</td>
<td>0.071</td>
</tr>
</tbody>
</table>

study are:

1. The optimal film cooling effectiveness occurs in the lower-left quadrant of the $DC - T_p$ design space, and at lower duty cycles ($DC = 0.14$) and higher frequencies ($St \sim 1$). During the off-period of coolant injection, flow adjustment near the delivery tube include ingestion of the mainstream flow into the tube and the propagation of this ingested front that pushes the coolant in the delivery tube to be squeezed out from the downstream edge of the hole. Thus coolant continues to spill out during the off-period, stays close to the surface, and provides improved coverage. For these cases a double-deck vortical structures were detected, with a hairpin vortex in the upper deck riding on top of a counter rotating vortex pair in the lower deck. The vertical velocity induced by each pair of vortices is small, leading to an attached coolant to the surface.

2. High duty cycles result in forming distinct vortex rings that are detached from the surface and lead to increased entrainment of the crossflow and thus poor coverage.

3. High pulsation periods causes the penetration of the crossflow to the delivery-tube trailing surface and causes a significant temperature rise in the delivery tube and an unfavorable effect on the film cooling effectiveness.

4. At lower pulsation periods, the ingested gas does not penetrate all the way to the trailing surface of the delivery tube, and does not cause an increase in the trailing wall temperature inside the tube.
5. Several local optima were found, rendering the gradient-based optimization strategies inadequate for the current problem.
Chapter 5
Uncertainty Quantification of Film Cooling Effectiveness

The goal of the current study is to investigate the effect of the uncertainty in the blowing ratio on the film cooling effectiveness. We present a computationally affordable strategy to investigate the effect of randomness in blowing ratio on the performance of film cooling. One of the main considerations in studying the effect of uncertainty in film cooling applications is the computational cost of solving such a problem, since to accurately capture the dynamics of the flow, high-fidelity numerical simulations that resolve the relevant spatial and temporal scales are required [5].

We combine DNS using spectral/hp finite element method and ME-gPC method as a strategy to discretize the parametric space. The film-cooling geometry includes a flat surface with a 35-degree inclined coolant delivery tube fed by a plenum. The outcome of this study will enable us to quantify the effect of randomness in blowing ratio on the surface temperature or cooling effectiveness.

5.1 Problem specification

We solve the incompressible Navier-Stokes equations for velocity and pressure along with the advection-diffusion equation for temperature. These equations in non-dimensional form are given by:

\[ \nabla \cdot \mathbf{u} = 0, \quad (5.1) \]
\[ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u}, \quad (5.2) \]
\[ \frac{\partial \theta}{\partial t} + (\mathbf{u} \cdot \nabla)\theta = \frac{1}{Re Pr} \nabla^2 \theta, \quad (5.3) \]

where \( \mathbf{u} = \mathbf{u}(x, t; \xi) \), \( p = p(x, t; \xi) \) and \( \theta = \theta(x, t; \xi) \) are non-dimensional Cartesian velocity vector, pressure and temperature respectively and \( \xi \) is the random variable representing
Figure 5.1: Schematic of jet in a crossflow.

blowing ratio. Prandtl number is denoted by $Pr$ and Reynolds number by $Re$ defined as $Re = U_\infty D/\nu$ with crossflow velocity, $U_\infty$, and jet diameter, $D$, as the characteristic velocity and length scale respectively. Temperature ($T$) is non-dimensionalized using

$$\theta(x, t; \xi) = (T(x, t; \xi) - T_c)/(T_h - T_c),$$

where $T_h$ is the hot gas temperature and $T_c$ is the coolant temperature.

The schematic of the domain and the boundary conditions used to solve equations 5.1-5.3 are shown in figure 5.1. The origin of the coordinate system is at the center of the jet exit hole. The geometry includes a 35-degree delivery tube with diameter $D$ and length of $3.5D$ originating from a plenum with the size of $7.9D \times 4D \times 3D$. The main domain, where
the coolant and free-stream interact, spans the volume of $18D \times 6D \times 3D$ in streamwise, wall-normal and spanwise directions respectively. The center of the jet exit is located $6D$ downstream of the inlet boundary of the domain. For inflow boundary condition, a laminar boundary layer profile with the boundary layer thickness of $\delta_{99\%}/D = 1$ and temperature of $\theta = 1$ are used. The laminar boundary layer is superposed with free-stream turbulence that is generated by the spectral synthesizer method introduced in [85]. The free-stream turbulence is assumed to be isotropic with the intensity of $0.5\%$ and length and time scale of $0.4D$ and $0.4D/U_\infty$ respectively. At the top boundary, free-stream condition is used with $u = (U_\infty, 0, 0)$ and $\theta = 1$. At the outflow boundary a zero-gradient condition is assumed for both velocity and temperature. In the spanwise direction a periodic boundary condition is enforced, mimicking a situation where an array of holes are arranged in the spanwise direction with the center-to-center distance of $3D$. For all of the wall boundaries, a no-slip boundary condition for velocity and adiabatic boundary condition for temperature is used. The Reynolds number, $Re_\infty = U_\infty D/\nu$, is 1500 throughout and the Prandtl number is $Pr = 0.71$.

At the bottom of the plenum, a spatially-uniform random vertical velocity of $u_2 = \alpha BR(\xi)$ is specified, where blowing ratio $BR$ is the ratio of the space-averaged jet velocity in the delivery tube ($U_j$) to the crossflow velocity ($U_\infty$) i.e. $BR = U_j/U_\infty$, and $\alpha$ accounts for the area ratio of the bottom of the plenum ($A_p$) to the normal cross-sectional area of the delivery tube, i.e. $\alpha = \pi D^2/4A_p$. The area scaling factor for the current geometry is $\alpha = 0.0331$. The random blowing ratio, denoted by $\xi \equiv BR$, is characterized by symmetrically truncating the tails of a Gaussian distribution around its mean. Considering a Gaussian distribution with the probability density function (PDF) of

$$\rho(\xi) = \frac{1}{\sqrt{2\pi}\sigma} \exp\left(-\frac{(\xi - m)^2}{2\sigma^2}\right),$$

with mean of $m$ and the standard deviation of $\sigma$, the PDF of the truncated Gaussian
distribution is given by:

\[
\mu(\xi) = \begin{cases} 
\rho(\xi)/\beta, & |\xi - m| \leq r, \\
0, & \text{otherwise}, 
\end{cases}
\]  
\hspace{1cm} (5.4)

where \( \beta \) is the scaling factor to enforce that \( \int_R \mu(\xi)d\xi = 1 \), and \( r \) is the distance from the mean beyond which \( \rho(\xi) \) is truncated. Note that both PDF's, \( \rho(\xi) \) and \( \mu(\xi) \), have the same mean value of \( m \), but their standard deviation values are different. In this study, we choose the mean value of \( m = 0.3 \), which is the blowing ratio at which the maximum film cooling effectiveness was observed according to the experimental measurements carried out by Bidan et al. [13] for a 35-degree inclined jet. The blowing ratio is allowed to have a range of variability in \([0, 0.6]\), which amounts to 100% variation around its mean \((r = 0.3)\). The standard deviation of the Gaussian distribution \( \rho(\xi) \) is 0.1 which results in the standard deviation of \( \sigma_{BR} = 0.098 \) for the truncated Gaussian distribution \( \mu(\xi) \). The probability density function of the random blowing ratio is shown in figure 5.2.

It should be noted that in representing the variation or randomness in the blowing ratio, the notion of a temporal scale or time varying blowing ratio is not introduced here. Rather, it is assumed that the scale associated with the random variation is decoupled and much larger than the time scales associated with the turbulent flow fluctuations. This could indeed be the case in the engine application since the coolant is supplied from the compressor stage upstream, through its own separate plumbing system and its inherent system-dynamics.

Since adiabatic conditions are used at the surface \( x_2 = 0 \), the surface temperature field \( \theta(x, t; \xi) \) at \( x_2 = 0 \) represents a measure of the surface cooling effectiveness by the coolant film, which is defined by:

\[
\eta(x_1; \xi) = \frac{1}{w} \int_{-w/2}^{w/2} (1 - \bar{\theta}(x; \xi)) \mid_{x_2=0} \, dx_3, 
\]  
\hspace{1cm} (5.5)
where $\eta(x_1; \xi)$ is the spanwise-averaged film cooling effectiveness, $w$ is the width of the cooled surface and the operator $\bar{()}$ represents a time-averaged quantity. Variations in blowing ratio ($\xi$) will lead to variations in temperature and equation 5.5 can be used to quantify this effect on film cooling effectiveness $\eta(x_1; \xi)$.

Further, a spatially-averaged (overall) film cooling effectiveness, denoted by $\tilde{\eta}(\xi)$, can be obtained from:

$$\tilde{\eta}(\xi) = \frac{1}{x_{1e} - x_{1s}} \int_{x_{1s}}^{x_{1e}} \eta(x_1; \xi) dx_1,$$

where $x_{1s}$ and $x_{1e}$ denote the beginning and the end of the streamwise interval over which $\tilde{\eta}(\xi)$ is calculated. In this study $x_{1s} = 1$ and $x_{1e} = 12$.

The objective of the current work is to investigate the effect of uncertainty of blowing ratio represented by the PDF shown in figure 5.2 on the measures of the film cooling effectiveness, $\eta(x_1; \xi)$ and $\tilde{\eta}(\xi)$, and the representative statistics associated with these measures.

### 5.2 Numerical algorithm

As it was demonstrated in the previous section, blowing ratio is a random number and adds another dimension, which is referred to as *parametric space*, to the governing equa-
tions. Note that solving the film cooling problem given by equations 5.1-5.3 for a constant deterministic blowing ratio requires performing a DNS simulation with several millions degree of freedom. Therefore adding another dimension requires an efficient discretization strategy to solve an already expensive problem.

In this section we explain the numerical algorithm that is used in the current study to solve equations 5.1-5.3.

5.2.1 Stochastic discretization

The details of discretization in the parametric space are given in [7]. Here the methodology is repeated with slight modifications for the sake of completeness.

- Decomposition in parametric space

We use ME-gPC introduced by Wan and Karniadakis [88] to discretize governing equations 5.1-5.3 in the one-dimensional parametric space which is formed by the random blowing ratio in the the range of $B = [0, 0.6]$. The parametric space $B$ is decomposed into $N_e$ non-overlapping elements denoted by: $B_1, B_2, B_3 \ldots, B_{N_e}$. Note that each element $B_e$ is a segment of the interval $[0, 0.6]$, and

$$B_e = [a_e, b_e],$$

where $a_e$ and $b_e$ are the beginning and the end of element $B_e$ respectively and since the elements are non-overlapping the end of one element is the beginning of the next element, i.e. $b_e = a_{e+1}$. The schematic of this decomposition is shown in figure 5.2. We introduce the indicator random variable:

$$I_{B_e}(\xi) = \begin{cases} 
1, & \xi \text{ belongs to } B_e, \\
0, & \text{otherwise.} 
\end{cases}$$

(5.7)

If a sample of blowing ratio, $\xi$, with the PDF given by equation 5.4, is drawn and the sample resides in the element $B_e$, the value of $I_{B_e}(\xi)$ is one and if the sample resides outside of the
element \( B_e \), the value of \( I_{B_e}(\xi) \) is zero. Using the law of total probability results in:

\[
Pr(\xi \leq q) = \sum_{e=1}^{N_e} Pr(\xi \leq q \mid I_{B_e} = 1)Pr(I_{B_e} = 1).
\]  (5.8)

Here \( Pr(\mathcal{E}) \) is the probability of event \( \mathcal{E} \), and \( Pr(\mathcal{E} \mid \mathcal{F}) \) is the conditional probability and represents the probability of event \( \mathcal{E} \) given that event \( \mathcal{F} \) has occurred. Using Bayes’ rule we reach at:

\[
\hat{\mu}_e(\xi \mid I_{B_e} = 1) = \frac{\mu(\xi)}{Pr(I_{B_e} = 1)},
\]  (5.9)

where \( \hat{\mu}_e(\xi \mid I_{B_e} = 1) \) is the conditional PDF with the condition of \( I_{B_e} = 1 \), which, by using equation 5.7, can be interpreted as: *if the random variable \( \xi \) resides in the element \( B_e \). The expectation of a generic function \( f(\xi) \) in the global domain \( B \) is obtained by:

\[
\mathbb{E}(f(\xi)) = \int_B f(\xi)\mu(\xi)d\xi.
\]  (5.10)

Using equations 5.8, 5.9 and 5.10, the expectation can be computed as:

\[
\mathbb{E}[f(\xi)] = \sum_{e=1}^{N_e} Pr(I_{B_e} = 1)\int_{B_e} f(\xi)\hat{\mu}_e(\xi \mid I_{B_e} = 1)d\xi.
\]  (5.11)

Equation 5.11 provides a basis for the calculation of statistical information from the elemental to the global level. The definition of the expectation motivates a natural weighted inner product with PDF as the weight. This follows:

\[
(f, g)_\mu := \mathbb{E}[f(\xi)g(\xi)], \quad \|f\|_\mu := (f, f)_\mu^{1/2},
\]  (5.12)

where \((.,.)_\mu\) is the inner product and \(\|.,\|_\mu\) is the norm with respect to the weight \(\mu(\xi)\).

Thus for two random functions of \( f(\xi) \) and \( g(\xi) \), the condition of \((f, g)_\mu = 0\) indicates orthogonality; a concept that is used in the next section to define an orthogonal basis for expansion of a generic function.
• **Stochastic collocation**

We consider a basis consisting of local polynomial chaos up to order $M$ in each element $B_e$ which are denoted by $\phi_i^e(\xi)$, where $i = 0, 1, \ldots, M$ is the polynomial order and $e = 1, 2, \ldots, Ne$ denotes the element number. A zero extension of polynomial $\phi_i^e(\xi)$ in region outside the element $B_e$ is assumed, i.e.:

$$\phi_i^e(\xi) = 0, \quad \xi \text{ outside of } B_e.$$

The polynomials $\phi_i^e(\xi)$ for $i = 0, \ldots M$ are mutually orthogonal with respect to the local PDF of $\hat{\mu}_e(\xi)$. In other words:

$$\int_{B_e} \phi_i^e(\xi) \phi_j^e(\xi) \hat{\mu}_e(\xi | I_{B_e} = 1) d\xi = (\gamma_i^e)^2 \delta_{ij}, \quad (5.13)$$

where $\gamma_i^e$ is the norm of the polynomial $\phi_i^e(\xi)$ and is given by:

$$\gamma_i^e = \int_{B_e} \phi_i^e(\xi)^2 \hat{\mu}_e(\xi | I_{B_e} = 1) d\xi. \quad (5.14)$$

These polynomials are constructed numerically using the procedure proposed by Wan and Karniadakis [88]. The polynomials $\phi_i^e(\xi)$ with $i = 0, 1, 2, \ldots$ form a hierarchical basis. Note that by considering the zero expansion of polynomial $\phi_i^e(\xi)$ outside of the element $B_e$, orthogonality in the global sense is also retained, i.e.:

$$\mathbb{E}[\phi_i^e(\xi) \phi_j^e(\xi)] = (\gamma_i^e)^2 \delta_{ij} \quad (5.15)$$

Now we consider a polynomial expansion for all time-averaged fields, which are shown by $\overline{()}$, as follows:

$$\overline{u}_M(x; \xi) = \sum_{e=1}^{Ne} \sum_{k=0}^{M} \overline{u}_k^e(x) \phi_k^e(\xi), \quad (5.16)$$
\[
\bar{y}_M(x; \xi) = \sum_{e=1}^{N_e} \sum_{k=0}^{M} \tilde{p}^e_k(x) \phi^e_k(\xi),
\]
(5.17)

\[
\bar{\sigma}_M(x; \xi) = \sum_{e=1}^{N_e} \sum_{k=0}^{M} \tilde{\theta}^e_k(x) \phi^e_k(\xi),
\]
(5.18)

where the subscript \( M \) represents the projection of the respective field onto the polynomial basis \( \phi^e(\xi) \), and \( \tilde{u}^e_k(x), \tilde{p}^e_k(x) \) and \( \tilde{\theta}^e_k(x) \) are the expansion coefficients. In the remaining of this section, in the interest of brevity, we carry out the formulation for temperature only, noting that the procedure can be similarly replicated for velocity and pressure. The expansion coefficients \( \tilde{\theta}^e_k(x) \) are obtained by employing Galerkin projection. This follows:

\[
\tilde{\theta}^e_k(x) = \frac{\mathbb{E}[\bar{y}_M(x; \xi) \phi^e_k(\xi)]}{\mathbb{E}[\phi^e_k(\xi)^2]}, \quad 0 \leq k \leq M, \quad 1 \leq e \leq N_e.
\]
(5.19)

To estimate the coefficient \( \tilde{u}^e_k(x) \), we use pseudo-spectral projection method [92] by using Gauss-quadrature rule which for a generic function \( f \) is defined as:

\[
\int_{B_e} f(\xi) \hat{\mu}_e(\xi | I_{B_e} = 1) d\xi \simeq \sum_{i=0}^{Q} w^e_i f(x; \xi^e_i),
\]
(5.20)

where \( \{\xi^e_i, w^e_i\}_{i=0}^{Q} \) are a set of \( Q + 1 \) Gauss-quadrature points and weights in element \( B_e \).

Note that the \((Q + 1)\)-point Gauss quadrature formulae is exact to evaluate \( \mathbb{E}[f] \) for any polynomial \( f \) up to degree \( 2Q + 1 \). By using the quadrature rule given by equation 5.20 and equation 5.14, the expansion coefficients for temperature given by equation 5.19 can be estimated as:

\[
\tilde{\theta}^e_k(x) \simeq \sum_{i=0}^{Q} w^e_i \theta(x; \xi^e_i) \phi^e_k(\xi^e_i) / \gamma^2_k, \quad 0 \leq k \leq M, \quad 1 \leq e \leq N_e.
\]
(5.21)

By replacing equation 5.21 in the polynomial expansion given by equation 5.18, a low-dimensional representation of random temperature is obtained which can serve as an off-line stochastic surrogate model whose evaluation is inexpensive.
• **Statistical information**

The equations 5.18 can serve as a stochastic surrogate model for temperature from which the statistical moments can be calculated.

1. **Mean:** the mean is simply calculated by taking $\mathbb{E}[\cdot]$ from both sides of equation 5.18:

$$
\mathbb{E}[\bar{\theta}_M(x; \xi)] = \sum_{e=1}^{N_e} \sum_{k=0}^{M} \hat{\theta}_e(x) \mathbb{E}[\phi_k^e(\xi)],
$$

(5.22)

2. **Variance:** using the orthogonality of the basis (equation 5.14), the variance is calculated from:

$$
\sigma^2_\theta(x) = \sum_{e=1}^{N_e} \sum_{k=0}^{M} \hat{\theta}_e(x)^2 \mathbb{E}[\phi_k^e(\xi)^2] - \sum_{e=1}^{N_e} \sum_{k=0}^{M} \hat{\theta}_e(x)^2 \mathbb{E}[\phi_k^e(\xi)]^2,
$$

(5.23)

where $\sigma_\theta(x)$ is the standard deviation of the temperature at point $x$ in space. The expansion for temperature given by equation 5.18 can accordingly be used to calculate the statistical moments for spanwise-averaged film cooling effectiveness, $\eta(x_1; \xi)$, and spatially-averaged film cooling effectiveness, $\tilde{\eta}(\xi)$, by using equations 5.5 and 5.6 respectively.

3. **Sensitivity:** the sensitivity of time-averaged temperature with respect to the random blowing ratio is defined as:

$$
S_\theta(x) = \mathbb{E} \left[ \frac{\partial \bar{\theta}_M(x; \xi)}{\partial \xi} \right] = \sum_{e=1}^{N_e} \sum_{k=0}^{M} \hat{\theta}_e(x) \mathbb{E} \left[ \frac{\partial \phi_k^e(\xi)}{\partial \xi} \right],
$$

(5.24)

where $S_\theta(x)$ denotes the sensitivity of time-averaged temperature with respect to random variation of blowing ratio. The issue of the discontinuity of the polynomial basis $\phi_k^e(\xi)$ at element boundaries is addressed in [7].

The statistical moments can be calculated for time-averaged velocity and pressure similarly.
Note that using equation 5.5, the expectation, variance and sensitivity for spanwise-averaged film cooling effectiveness become:

\[
\mathbb{E}[\eta(x_1; \xi)] = \frac{1}{w} \int_{-w/2}^{w/2} \left(1 - \mathbb{E}[\bar{\theta}_M(x; \xi)]\right) |_{x_2=0} \, dx_3, \tag{5.25}
\]

\[
\sigma^2_\eta(x_1) = \frac{1}{w} \int_{-w/2}^{w/2} \sigma^2_\theta(x) \, |_{x_2=0} \, dx_3, \tag{5.26}
\]

\[
S_\eta(x_1) = \frac{1}{w} \int_{-w/2}^{w/2} S_\theta(x) \, |_{x_2=0} \, dx_3. \tag{5.27}
\]

- Discretization details

In the current study, we use five elements, i.e. \( Ne = 5 \). The specification of the domain decomposition is given in table 5.1. The decomposition provides a set of non-overlapping elements that covers the range of \( BR \) in the range of \([0, 0.6]\). The elements are refined at \( BR = 0 \) where \( BR = 0 \) represents the condition that the jet is off. By increasing blowing ratio from zero large gradients of velocity, pressure and temperature are expected, which implies that local refinement may be necessary. This is consistent with the properties of the ME-gPC method where a local-refinement strategy can be adopted by either increasing the polynomial order locally (known as \( p \)-refinement), or refining the element size (known as \( h \)-refinement); the latter of which is adopted in this study. On the other hand, in gPC method, increasing the polynomial order globally is the only refinement strategy which may not be efficient if low regularity or sharp transition exists locally.

We choose polynomial order \( M = 3 \) within each element. These polynomials for the first and fourth elements are shown in figure 5.3. Note that the polynomial basis are orthogonal with respect to the local PDF of \( \tilde{\mu}_e(\xi \mid I_{B_e} = 1) \) in each element and as a result the polynomials of the same degree are different from one element to another. We use 4 Gauss quadrature points \( (Q = 3) \) which results in a degree of exactness \( 2Q + 1 = 7 \) when evaluating \( \mathbb{E}[f] \) in each element. In total, \( Ne \times (Q + 1) \) times sampling of the governing equations 5.1-5.3 is required. Therefore for the given parameters, 20 DNS simulations are
Table 5.1: Multi-element decomposition of the random space

<table>
<thead>
<tr>
<th>Element</th>
<th>$a_e$</th>
<th>$b_e$</th>
<th>$Pr(I_{B_e} = 1)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$B_1$</td>
<td>0.00</td>
<td>0.05</td>
<td>0.0049</td>
</tr>
<tr>
<td>$B_2$</td>
<td>0.05</td>
<td>0.15</td>
<td>0.0607</td>
</tr>
<tr>
<td>$B_3$</td>
<td>0.15</td>
<td>0.30</td>
<td>0.4343</td>
</tr>
<tr>
<td>$B_4$</td>
<td>0.30</td>
<td>0.45</td>
<td>0.4343</td>
</tr>
<tr>
<td>$B_5$</td>
<td>0.45</td>
<td>0.60</td>
<td>0.0656</td>
</tr>
</tbody>
</table>

performed.

5.2.2 Discretization in space and time

In all the cases in this study we perform direct numerical simulation on the jet in crossflow with the schematic of the problem shown in figure 5.1. For discretization in space, we use spectral element method with hexahedral element implemented in $N\epsilon\kappa\alpha\tau$ [89]. A third-order semi-implicit fractional step method is used to advance the governing equations in time. For more details on the method see section 2.2.

A three-dimensional view of the computational grid that is used in this study is shown in figure 5.4(a). To generate the grid, a total number of 2226 quadrilateral elements were first generated in $x_1-x_3$-plane at the jet exit elevation, out of which 372 elements belonged to the jet exit. A closer view of this grid near the jet exit is shown in figure 5.4(b). The two-dimensional grid was then swept along the $x_2$ direction generating 17 parallel layers of grid with increasing height away from the wall. The first-layer elements have a height of $\Delta x_2 = 0.05D$ and large elements with the height of $3D$ were used for the topmost layer. Similarly, within the pipe the grid at the jet exit was swept with 24 layers along the axis of the tube with the height of $\Delta x_2 = 0.07D$ for the first and last layer. An analogous strategy to that of the main domain was used for generating grid in the plenum where first a quadrilateral grid with 720 elements were generated at the top $x_1-x_3$-plane. These elements were then swept downward to the plenum inlet creating 11 layers with $\Delta x_2 = 0.06D$ at the topmost layer and $\Delta x_2 = 1.2D$ at the the bottommost layer. The final
grid obtained had 52464 elements. We use polynomial with spectral order of four which translates into $(4 + 1)^3 = 125$ points within each element, resulting in approximately a total of 6.5 millions points. The same grid is used in all simulations and the grid spacing in the wall coordinate $x_i^+ = u_\tau x_i/\nu$, where $u_\tau = \sqrt{\nu(\partial u_1/\partial x_2)_{wall}}$ is the average friction velocity for the highest blowing ratio ($BR = 0.5841$), is related to the regular spacing by $\Delta x_i^+ = 375\Delta x_i$. Given that polynomial with spectral order of four is used within each element, the accuracy is comparable with Fourier spectral method with two complete modes resolved in each element. On the other hand a finite difference approximation would require 5 points to capture two complete Fourier modes. Therefore the resolution in the wall-normal direction of the first-layer element in the main domain in a finite difference spirit is...
Figure 5.4: Unstructured hexahedral grid; (a) three-dimensional view; (b) $x_1 - x_3$ view of the grid in the vicinity of the jet exit, black lines: element boundaries; gray lines: Gauss-Lobatto-Legendre quadrature grid with spectral order $m = 4$.

$\Delta x_2^+ = 0.05 \times 375/4 = 4.69$. However note that the spectral element with polynomial order of $P$ is significantly more accurate than a $(P + 1)$-point-finite-difference approximation. For a more detailed discussion on this see [50]. In the same way, the resolution of the grid in the main domain in the streamwise direction is $7.5 \leq \Delta x_1^+ \leq 20$ and in the spanwise direction $7.5 \leq \Delta x_3^+ \leq 14$. The spectrum of the kinetic energy $E = (u_1^2 + u_2^2 + u_3^2)^{1/2}/2$ at point $x_1 = 2$, $x_2 = 1$ and $x_3 = 0$ for the highest blowing ratio considered in this study is shown in figure 5.5. The Kolmogorov inertial range with the slope of $-5/3$ is observed.
Figure 5.5: Spectrum of the energy of the velocity signal at location $x_1 = 2$, $x_2 = 1$ and $x_3 = 0$ with blowing ratio of $BR = 0.5841$.

5.3 Results

5.3.1 Effect of blowing ratio on film cooling

We performed twenty direct numerical simulations in total, at Gauss-quadrature points in the parametric space, where each quadrature point represents a unique blowing ratio. For each simulation, time-averaged quantities are collected and used in equation 5.21 to compute the expansion coefficients. Finally the stochastic response of the system is constructed using equations 5.16-5.18 from which all the statistical information can be directly extracted.

Figure 5.6 shows the instantaneous temperature field at the mid-plane ($x_3 = 0$) at six quadrature points in the parametric space. At $BR = 0.004$, a significant ingestion of the crossflow gas into the delivery tube can be seen. As the blowing ratio increases ($BR = 0.1442$), the coverage extends to a larger portion of the surface downstream of the jet, improving the spatially-averaged film cooling effectiveness. This trend continues up to $BR = 0.2902$ where the shear layer tends to become unstable around $x_1 = 6$ as
evidenced by the unsteadiness in the time sequence of the data. At $BR = 0.3956$, a clear roll-up of the shear layer can be seen which leads to the formation of hairpin vortices that shed periodically after $x_1 \geq 6$. As blowing ratio further increases, the coolant jet detaches from the surface, leading to the reduction of the coverage in the near-hole region. At $BR = 0.4891$ the roll-up of the shear layer occurs closer to the hole and due to the separation of the coolant from the surface, film cooling effectiveness continues to decrease. At $BR = 0.5841$, the jet is fully detached and the crossflow then penetrates in the region behind the jet body and causes a poor coverage of the surface.

Figure 5.7 shows the time-averaged temperature on the cooled surface for all quadrature points. It is clear that as blowing ratio increases from the small value of $BR = 0.004$ to $BR = 0.2044$, the film cooling effectiveness improves everywhere on the surface. However further increase in blowing ratio ($BR = 0.2538$) results in reduction of the film cooling effectiveness in the vicinity of the jet exit due to the jet liftoff, but further downstream ($x_1 > 5$) the effectiveness still increases which is caused by the re-attachment of the coolant jet. The spanwise coverage also increases modestly up to higher blowing ratios ($BR \geq 0.4891$), after which the deterioration of the coverage is more pronounced immediately downstream of the jet exit ($x_1 < 2$) and less strongly further downstream ($x_1 > 5$).

### 5.3.2 Spatially-averaged film cooling effectiveness

The previous observations from figures 5.6 and 5.7 indicate that an optimal blowing ratio exists. In figure 5.8 the spatially-averaged film cooling effectiveness, $\tilde{\eta}(\xi)$, versus blowing ratio is shown. The optimal blowing ratio is approximately at $BR_{opt} = 0.3$, which is in agreement with the experimental study carried out by Bidan et al. [13]. The projection of $\tilde{\eta}(\xi)$ to different polynomial orders ranging from $M = 0$ to $M = 3$ is also shown. The fast convergence of $\tilde{\eta}(\xi)$ can be seen where the difference between the projection with order of $M = 2$ and $M = 3$ (red and green lines in Figure 5.8) is negligible. Note that the polynomial $\phi_i^e(\xi)$ are discontinuous across elements, however the discontinuity at the element interfaces is of measure zero (i.e., negligible) with respect to the norm given by
Figure 5.6: Instantaneous temperature surface in the mid-plane ($x_3 = 0$) for different blowing ratios.
Figure 5.7: Time-averaged temperature contours for quadrature points on cooled surface ($x_2 = 0$). In this figure simulations for all Gauss quadrature points are shown.

equation 5.12. One could have chosen Gauss-Lobatto-Legendre quadrature points where the end points of the elements are included and thus the continuity across the elements is ensured, however in that case the degree of exactness for $(Q + 1)$ quadrature points is $2Q - 1$ as opposed to the degree of exactness of $2Q + 1$ for the Gauss quadrature points used in this study, resulting in larger amount of error overall for the Gauss-Lobatto-Legendre quadrature points. Nevertheless for a fixed elemental decomposition, the current choice of quadrature points gives the best polynomial approximation when the error is measured in the norm given by equation 5.12. For a more detailed discussion in this regard see [88].

Mathematically, the relationship between the PDF of the random blowing ratio $\mu(\xi)$
and the PDF of film cooling effectiveness $\rho(\eta)$ is determined by:

$$\rho(\eta) = \mu(\xi) / \left| \frac{d\eta}{d\xi} \right|,$$

(5.28)

where we assume that $\eta(\xi)$ is monotonic for simplicity. Equation 5.28 has two implications:

1. The regions in which blowing ratio has high probability of occurrence, increases the likelihood of the film cooling effectiveness occurrence at values of $\eta(\xi)$ associated with those blowing ratios.

2. The PDF of the film cooling effectiveness, $\rho(\eta)$, is inversely proportional to the slope of $\eta$ vs. $\xi$. In other words, the regions where film cooling effectiveness is not sensitive to the variation of blowing ratio, i.e. low $d\eta/d\xi$, tend to increase the likelihood of occurrence of values of $\eta$ corresponding to those blowing ratios.
As it is clear from figure 5.8, $\tilde{\eta}(\xi)$ is not a monotonic function of $\xi$, and therefore equation 5.28 cannot be used to calculate the PDF of $\tilde{\eta}(\xi)$. Instead, we use Monte Carlo method where $\xi$ is sampled from its distribution, the truncated Gaussian $\mu(\xi)$ with the mean of 0.3 and standard deviation of $\sigma_{BR} = 0.098$. This is achieved by first drawing samples from Gaussian distribution $\rho(\xi)$ with the mean of 0.3 and the standard deviation of 0.1. Then the samples $\xi$ that are outside the region of $[0, 0.6]$, are rejected. We draw twenty million samples from $\xi$’s distribution. The histogram plot of the samples and the exact PDF are shown in figure 5.9 and negligible difference between the two is observed, indicating that the sample size is sufficiently large. Next $\tilde{\eta}(\xi)$ is evaluated at all sampled $\xi$’s, creating a collection of twenty million samples of $\tilde{\eta}(\xi)$, whose PDF is shown in figure 5.10. Note that sampling from the $\xi$’s distribution and evaluating $\tilde{\eta}(\xi)$ at these samples are carried out as post-processing and it takes only seconds to perform the calculations.

The maximum value of $\tilde{\eta}(\xi)$ is 0.254 and all other values for $\tilde{\eta}(\xi)$ are below this peak.
number. Therefore, the probability of $\eta(\xi)$ beyond this peak is zero, which is reflected by the sudden drop of the value of $\rho(\eta)$ to zero for $\eta(\xi) > 0.254$. The value of $\rho(\eta)$ becomes unbounded at $\eta = 0.251$, since $d\eta/d\xi = 0$ at this point. There is a second smaller peak at $\eta$ of 0.246 which is related to the near-zero slope of $\eta(\xi)$ at this value as shown in figure 5.10. Two square-shape jumps in the profile of $\rho(\eta)$ in the region of 0.215 < $\eta$ < 0.225 are triggered by the small discontinuity of $\eta(\xi)$ at elemental boundaries (discussed in the preceding paragraph).

Once the film cooling effectiveness response is obtained all statistical moments can be computed. The expected value for the spatially-averaged film cooling effectiveness is $\mathbb{E}[\eta] = 0.238$ and is shown by a vertical dashed line in figure 5.10. It is important to note that this expected value is lower than the optimal value of effectiveness of 0.25 at the design $BR$ of 0.3. This deterioration in cooling performance is not surprising since it is linked to the variation in the blowing ratio around the 0.3 value, and cooling performance worsens on
either side of the 0.3 value. The standard deviation of the spatially-averaged film cooling effectiveness is $\sigma_{\tilde{\eta}} = 0.022$ which is equivalent to $\pm 9.4\%$ variation with respect to the expected value. This implies an overall cooling effectiveness range of 0.216-0.26. In view of the blowing ratio variations that are naturally present in the gas turbine environment, the expected value $E[\tilde{\eta}]$ and the range of the cooling effectiveness around $E[\tilde{\eta}]$ are of significant value to the turbine designer because it allows for the incorporation of realistic effects such as the blowing ratio variations.

5.3.3 Spanwise-averaged film cooling effectiveness

In figure 5.11, the spanwise-averaged film cooling effectiveness at six streamwise stations are shown. The film cooling effectiveness at $x_1 = 2$ increases from $BR = 0$ and reaches the maximum of $\eta = 0.38$ at $BR = 0.18$. Further increase in blowing ratio leads to the jet liftoff and a decrease in $\eta$ is observed. The slope $d\eta/d\xi$ decreases as the maximum $\eta$ value is reached which according to equation 5.28, causes the sharp increase of $\rho(\eta)$ as it is shown in figure 5.12(a). At the maximum of $\eta$ ($\eta = 0.38$), $d\eta/d\xi = 0$ and therefore $\rho(\eta)$ becomes
unbounded. As the blowing ratio increases, the magnitude of slope of $\eta(\xi)$ increases as well, which contributes to the decrease in $\rho(\eta)$. However the likelihood of blowing ratio residing in this region increases (large $\mu(\xi)$) which prohibits a fast decay of $\rho(\eta)$. As a result a relatively large spread of $\rho(\eta)$ around its mean is observed, which is responsible for
large values of the standard deviation. Physically, large amount of the standard deviation implies significant fluctuations in the cooling effectiveness values despite the high cooling effectiveness values of 0.38 at this region. This is further illustrated in figure 5.13, where at \( x_1 = 2 \), the expected value \( \mathbb{E}[\eta] = 0.316 \) and the standard deviation is 0.058. At \( x_1 = 3 \) the maximum of \( \eta \) slightly shifts to a higher value of \( \xi \) and smaller slopes are observed compared to \( \eta(\xi) \) at \( x_1 = 2 \), leading to a more concentrated distribution around its mean as it is shown in figure 5.12(b). At \( x_1 = 4 \) the maximum of \( \eta \) occurs at higher blowing ratio of \( BR = 0.23 \). This overlap of the low-slope region near the \( \eta \)-maximum with the region of high probability for blowing ratio tends to centralize \( \rho(\eta) \) in the overlapped region and thus it decreases the spread of the PDF as it is shown in figure 5.12(c). As a consequence the variance at this \( x_1 \) decreases. This trend continues up to \( x_1 = 6 \), where the width of the low-slope region widens and also the maximum of \( \eta \) nearly coincides with the maximum of \( \mu(\xi) \) at \( \xi = 0.3 \). This gives rise to the most concentrated \( \rho(\eta) \) among the four cases considered here, as it is shown in figure 5.12(d), and therefore the least amount of the standard deviation is observed at \( x_1 = 6 \). This implies that at this downstream location, despite the variability in the upstream BR, its impact on the cooling effectiveness variations is muted relative to the upstream locations. At \( x_1 = 8 \) the maximum of \( \eta \) slightly shifts to higher blowing ratio (\( BR = 0.4 \)), and downstream of the peak in the \( \mu(\xi) \) curve. However the relatively wide region of flat \( \eta \) creates a low-sensitivity relationship between \( \eta \) and \( BR \). This plateau region extends nearly up to \( BR = 0.3 \) which results in a fairly concentrated \( \rho(\eta) \) and small value of standard deviation (see figure 5.12(e)). Further downstream at \( x_1 = 10 \), the average of \( d\eta/d\xi \) slightly increases which leads to a small increase in the standard deviation compared to \( x_1 = 8 \). In figure 5.13 the optimal film cooling effectiveness \( (\eta_{\text{opt}}(x_1) \equiv \eta(x_1; BR_{\text{opt}} = 0.3) \) and the expectation \( (\mathbb{E}[\eta(x_1; \xi)]) \) of the spanwise-averaged film cooling effectiveness are shown. The streamwise axis is extended to include the jet hole and its upstream segment. The optimal effectiveness has higher values everywhere downstream of the jet. The vertical bars show the range of \( \mathbb{E}[\eta(x_1; \xi)] \pm \sigma_\eta(x_1) \)
Figure 5.13: Uncertainty in the spanwise-averaged film cooling effectiveness $\eta(x_1;\xi)$.

Figure 5.14: Standard deviation of temperature, $\sigma_\theta(x)$ on the cooled surface of $x_2 = 0$.

where $\sigma_\eta(x_1)$ is the standard variation of $\eta(x_1;\xi)$. It is clear that the randomness of the blowing ratio does not have any effect on the region upstream of the hole and therefore the standard deviation is zero in this segment. As it was demonstrated above, the random blowing ratio has the maximum impact near the jet trailing edge. The standard deviation decreases after $x_1 \geq 2.5$ and reaches a minimum near $x_1 = 6$ and then increases again till $x_1 \simeq 9.5$ and remains almost unchanged thereafter. This behavior is quantified in figures 5.14 and 5.15. In figure 5.14 the contour plot of the standard deviation of temperature,
\( \sigma_{\theta}(x) \), is shown on the cooled surface of \( x_2 = 0 \). Note that the standard deviation of time-averaged temperature is also equivalent to the standard deviation of the local film cooling effectiveness, since \( \bar{\theta}(x_1, 0, x_3; \xi) = 1 - \eta(x_1, x_3; \xi) \). The region immediately downstream of the jet hole has the largest amount of variability in temperature of \( \sigma_{\theta} \simeq 0.15 \). Figure 5.15 shows the sensitivity and standard deviation of spanwise-averaged film cooling effectiveness.

The highest magnitude of the sensitivity is reached at \( x_1 = 2 \), which also corresponds to maximum value of standard deviation, both re-affirming the existence of the high-sensitivity region downstream of the jet hole. It should be noted that \( S_\eta(x_1) \), is a measure of the overall sensitivity of \( \eta(x_1; \xi) \) with respect to blowing ratio. As it can be seen in figure 5.11, \( \eta(x_1; \xi) \) at \( x_1 = 2 \) has a large portion of negative slope coinciding with the high-probability region. This leads to the negative sign of sensitivity in this region. The magnitude of sensitivity decreases at \( x_1 > 2 \) and becomes zeros at \( x_1 \simeq 5 \). At \( x_1 \simeq 5 \) (although not shown in figure 5.11), the maximum of \( \eta(x_1; \xi) \) roughly occurs at the design point \( \xi = 0.3 \). This leads to a balanced distribution of positive- and negative-slope regions around \( \xi = 0.3 \) with a symmetric probability distribution around it. This means that the random \( \eta(x_1; \xi) \) at \( x_1 \simeq 5 \) has equal samples of positive and negative slope of \( d\eta/d\xi \) that have resulted in their cancellation and thus zero sensitivity is resulted. Note that the standard deviation reaches its minimum at roughly the same location of \( x_1 \simeq 5 \). At \( x_1 \geq 5 \), the positive-slope region outweighs the negative-slope region (see figure 5.11) resulting in positive sensitivities. The magnitude of the sensitivity also increases as the maximum of the \( \eta(x_1; \xi) \) occurs at higher blowing ratios, resulting in a smaller negative-slope region. The value of standard deviation also increases due to the increasing offset between the maximum of \( \eta(x_1; \xi) \) and \( \mu(\xi) \).

### 5.4 Concluding remarks

In this study an efficient numerical algorithm for quantifying the effect of uncertainty of blowing ratio on film cooling performance has been presented. The geometry of the problem includes a plenum and a 35-degree inclined delivery tube. The blowing ratio is a random variable associated with a truncated Gaussian distribution with the mean of 0.3
and the standard deviation of 0.098. A multi-element general polynomial chaos is utilized to discretize the parametric space into non-overlapping elements and an orthogonal polynomial expansion within each element. A pseudo-spectral method has been used to find the expansion coefficients in a non-intrusive manner by sampling the governing equations at Gauss-quadrature points. The spectral element method has been used to perform direct numerical simulation at each quadrature point. The findings of this study can be summarized as:

1. A probabilistic framework to quantify the effect of randomness in the blowing ratio on film cooling effectiveness is presented.

2. Fast convergence of the the general polynomial chaos combined with capability of local refinement offered by ME-gPC method is as an effective strategy to evaluate the effect of uncertainty or randomness in the blowing ratio on the cooling performance.
The total cost of computation is equivalent of performing twenty independent direct numerical simulations that can run concurrently on segmented parallel clusters.

3. Using the approach in the current study, all the statistical information of the time-averaged quantities, such as probability density function, expectation and variance, sensitivity, etc can be calculated. This information can be integrated into a Bayesian approach for probabilistic design, and future studies at higher Reynolds number will explore these extensions into probabilistic design.

4. The laterally-averaged cooling effectiveness has its maximum value closer to the trailing edge of the coolant hole. This maximum value occurs at a lower blowing ratio, and is associated with the largest variance.

5. The expected values are lower than the design cooling effectiveness at the design $BR$ of 0.3 and reflect the influence of the blowing ratio randomness.

6. The standard deviation of the surface temperature indicates the randomness in blowing ratio causes the highest amount of temperature variation ($\sigma_\theta \simeq 0.15$) in the region extending from trailing edge of the exit hole to four diameters downstream.

7. The most and least sensitive regions to the variation in blowing ratio occur at two jet diameters and five jet diameters downstream of the hole respectively.
Chapter 6
A Finite difference Method for Solving Incompressible Navier-Stokes Equations

In the current study, we extend the semi-staggered approach to solve time-dependent Navier-Stokes equations on three dimensional curvilinear grids. As noted in chapter 1, the current semi-staggered approaches can be susceptible to oscillatory pressure modes. The motivation of the current study stems from the fact that the existing numerical methods for solving the time-dependent incompressible Navier-Stokes equations on general curvilinear grids do not jointly meet the following attributes, all of which are essential for an accurate and efficient numerical simulation: (1) spurious pressure eigen-modes are absent, (2) the discrete continuity is satisfied exactly, (3) Christoffel symbol is avoided, (4) irregular half-cells are absent in the velocity grids, and finally (5) ad hoc pressure boundary condition is not required. The current semi-staggered approaches do not satisfy conditions (1) [29, 69], and staggered grid approaches also do not satisfy condition (4) and collocated approaches fail to satisfy condition (2). In the current work we demonstrate that our method meets all of the above-mentioned attributes.

6.1 Governing equations

We consider the Navier-Stokes equations and the energy equation using Boussinesq approximation that govern the time-dependent incompressible flow. The non-dimensional form of these equations in index notation are:

\[
\frac{\partial u_i}{\partial t} = 0 \quad \text{in } \Omega. \tag{6.1}
\]

\[
\frac{\partial u_i}{\partial t} + \frac{\partial (u_j u_i)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} \right) + \frac{Gr}{Re^2} \theta e_i \quad \text{in } \Omega, \tag{6.2}
\]

\[
\frac{\partial \theta_i}{\partial t} + \frac{\partial (u_j \theta)}{\partial x_j} = \frac{1}{Re Pr} \frac{\partial}{\partial x_j} \left( \frac{\partial \theta}{\partial x_j} \right) \quad \text{in } \Omega, \tag{6.3}
\]
where \( \text{Re} \), \( \text{Pr} \) and \( \text{Gr} \) are relevant non-dimensional parameters, and \( \mathbf{e} \) is a vector in the direction of buoyancy force. We solve equations 6.1-6.3 on complex geometries using the non-orthogonal body-fitted grid. The Cartesian coordinates are mapped to the curvilinear coordinates by the transformation \( \mathbf{\xi} = \mathbf{\xi}(\mathbf{x}) \) characterized by the Jacobian of the transformation \( \mathbb{J} = \frac{\partial \mathbf{x}}{\partial \mathbf{\xi}} \). This follows [26]:

\[
\frac{\partial u_i}{\partial t} + J^{-1} \frac{\partial (U^j u_i)}{\partial \xi^j} = -\frac{\partial p}{\partial \xi^j} \frac{\partial \xi^j}{\partial x_i} + J^{-1} \frac{1}{Re} \frac{\partial}{\partial \xi^j} (B^{jk} \frac{\partial u_i}{\partial \xi^k}), \tag{6.4}
\]

\[
J^{-1} \frac{\partial U^i}{\partial \xi^i} = 0, \tag{6.5}
\]

where

\[
B^{ij} = J \frac{\partial \xi^i}{\partial x_k} \frac{\partial \xi^j}{\partial x_k}. \tag{6.6}
\]

The contra-variant flux vector and is related to the Cartesian velocity by:

\[
U^j = \beta^j \cdot \mathbf{u}, \tag{6.7}
\]

where the vector \( \beta^j \) is orthogonal to \( \xi^j \)-constant face and is given by:

\[
\beta^j_i = J \frac{\partial \xi^j}{\partial x_i}, \tag{6.8}
\]

and using the inverse transformation the Cartesian velocity is related to the contra-variant flux by:

\[
u_k = U^j \frac{\partial x_k}{\partial \xi^j}. \tag{6.9}
\]

Note that in the transformed equations 6.4-6.5, only Cartesian coordinates are mapped to curvilinear coordinates and the dependent variable is still the Cartesian velocity \( \mathbf{u} \). This allows us to avoid using the Christoffel symbol that appears in the fully transformed equations ([53, 96]).

6.2 Numerical algorithm

In the current study we present an improved finite difference algorithm to solve the time-dependent incompressible Navier-Stokes equations on curvilinear grids. To advance in
time, we use the fractional step method which consists of two steps, (1) calculation of the intermediate velocity without the pressure gradient and (2) the projection step that adds the pressure gradient. In between these steps, a pressure Poisson equation has to be solved to determine the pressure field. Once the velocity field is obtained, the energy equation is advanced in time to obtain temperature at new time level.

Our method primarily takes advantage of the semi-staggered grid structure (figure 1.3(d)) in which all three components of the provisional momentum equations and temperature are discretized at grid vertices, thereby avoiding the need to interpolate the velocity components across the three different logical grids as in the staggered grid. Moreover the discretization of the convective and diffusive terms at grid points adjacent to the domain boundary is facilitated without requiring to use half- or ghost-cell treatment, which are generally required in the staggered grid layout. However our method departs from the semi-staggered grid in the projection step, where we adopt a staggered grid discretization for the Poisson-Neumann equation and thus avoid any spurious pressure modes. Therefore, our new approach may be viewed as a hybrid of the semi-staggered and staggered approaches that eliminates the major disadvantages of each these methods when a curvilinear grid is used.
6.2.1 Metrics and discrete operators

A linear mapping is used to transform the curvilinear grid from the physical space to a uniform grid in the computational space (see figure 6.1). The mapping is chosen such that the mesh size in the computational domain is $\Delta \xi^1 = \Delta \xi^2 = \Delta \xi^3 = 1$. Using this choice of mesh size, $|\langle \beta^i \rangle| = \sqrt{(\beta^i_1)^2 + (\beta^i_2)^2 + (\beta^i_3)^2}$ represents the area of $i$-constant face, and $U^i$ represents the volumetric flux passing through the $i$-constant face. We adapt the convention that the cell vertex is indexed as $(i, j, k)$, cell center as $(i+1/2, j+1/2, k+1/2)$, $\xi^1$-constant face is indexed as $(i, j+1/2, k+1/2)$ and $\xi^2$- and $\xi^3$-constant faces are indexed similarly (see figure 7.1(d)).

The metric terms are discretized using a second-order central finite difference scheme. In order to ensure the surface conservation for a closed cell, a double grid is first constructed by interpolating the coordinates of the grid points to the desired locations, and then using the interpolated coordinates to calculate the metric terms[87]. For the metric terms at the physical boundaries, a one-sided second-order discretization is used.

We introduce the finite difference operators that are frequently used throughout this text. The discrete averaging and first-order-derivative operators, in the generic sense, are:

$$\left. \overline{\psi} \right|_{\xi^*} := \left. \psi \right|_{\xi^*} + \mathcal{O}(\Delta \xi^\alpha), \quad (6.10)$$

$$\left. \delta \psi \overline{\xi^i} \right|_{\xi^*} := \left. \frac{\partial \psi}{\partial \xi^i} \right|_{\xi^*} + \mathcal{O}(\Delta \xi^\alpha), \quad (6.11)$$

where $\psi$ is a general scalar function and $\xi^* = \{\xi^*^1, \xi^*^2, \xi^*^3\}$ denotes the point at which the operators are evaluated, and $\alpha$ is the order of accuracy of the discrete approximation. More discrete operators are defined in table 6.1, as they will be required for the discretization of equations 6.4-6.5. Table 6.1 also shows the specific operands on which the operators act. More specifically, we introduce the following second-order accurate operators:

$$\left. \overline{\psi} \right|_{i,j,k} := \frac{\psi(i+1/2, j, k) + \psi(i-1/2, j, k)}{2}, \quad (6.12)$$

92
Table 6.1: Discrete differential operators.

<table>
<thead>
<tr>
<th>Operator definition</th>
<th>Descriptor</th>
<th>Operand</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\mathcal{N}(v) := \frac{1}{J} \frac{\delta(U^j v_i)}{\delta \xi^j}$</td>
<td>convective</td>
<td>co-variant vector field $v$</td>
</tr>
<tr>
<td>$\mathcal{L}(v) := \frac{1}{J} \frac{\delta}{\delta \xi^j}(B_{jk} \frac{\delta v_i}{\delta \xi^k})$</td>
<td>diffusive</td>
<td>co-variant vector field $v$</td>
</tr>
<tr>
<td>$\mathcal{D}(V) := \frac{1}{J} \frac{\delta V_i}{\delta \xi^i}$</td>
<td>divergence</td>
<td>contra-variant vector field $V$</td>
</tr>
<tr>
<td>$\mathcal{G}_x(\psi) := \frac{1}{J} \beta_i \frac{\delta \psi}{\delta \xi^i}$</td>
<td>co-variant gradient</td>
<td>scalar field $\psi$</td>
</tr>
<tr>
<td>$\mathcal{G}_\xi(\psi) := B^{ij} \frac{\delta \psi}{\delta \xi^j}$</td>
<td>contra-variant gradient</td>
<td>scalar field $\psi$</td>
</tr>
</tbody>
</table>

$$\frac{\delta u^i}{\delta \xi^j}|_{i,j,k} := \frac{\psi(i + n/2, j, k) - \psi(i - n/2, j, k)}{2}. \quad (6.13)$$

The discrete operators 6.12 and 6.13 are defined similarly in other directions.

6.2.2 Provisional momentum equation

The time integration method used in this study is based on a semi-implicit fractional step method [52], where first the provisional momentum equations are integrated in time to obtain the intermediate velocity $\tilde{u}$, and in the second step a Poisson-Neumann problem is solved to obtain the projection function $\phi$ that projects the intermediate velocity onto a divergence-free velocity field. The convective and diffusive terms are advanced using the second-order Adams-Bashforth method [75] and Crank-Nicolson method respectively. It follows that:

$$\frac{\tilde{u} - u^n}{\Delta t}|_{i,j,k} = \frac{1}{2Re} (\mathcal{L}(\tilde{u}) + \mathcal{L}(u^n)) - \frac{1}{2}(3\mathcal{N}(u^n) - \mathcal{N}(u^{n-1})) + \frac{Gr}{2Re^2}(3\theta^n - \theta^{n-1})e, \quad (6.14)$$

$$\tilde{u}|_\Gamma = u^{n+1}|_\Gamma. \quad (6.15)$$
The diffusive terms are discretized using the second-order central finite difference scheme. A second-order derivative is approximated as:

$$\frac{\partial}{\partial \xi_1}(B^{11} \frac{\partial u}{\partial \xi_1})_{i,j,k} = \frac{\delta_1}{\delta \xi_1}(B^{11} \delta_1 u)_{i,j,k} + O((\Delta \xi_1)^2), \quad (6.16)$$

and a cross-derivative as:

$$\frac{\partial}{\partial \xi_1}(B^{12} \frac{\partial u}{\partial \xi_2})_{i,j,k} = \frac{\delta_2}{\delta \xi_1}(B^{12} \delta_2 u)_{i,j,k} + O((\Delta \xi_1)^2, (\Delta \xi_2)^2). \quad (6.17)$$

Convective terms are discretized using a third order biased upwind scheme [72] which for $f := Uu$ is given by:

$$\frac{\delta f}{\delta \xi_1} = \begin{cases} 
\left(2f_{i+1,j,k} + 3f_{i,j,k} - 6f_{i-1,j,k} + f_{i-2,j,k}\right)/6, & U_{i,j,k} \geq 0, \\
\left(f_{i+2,j,k} - 6f_{i+1,j,k} + 3f_{i,j,k} + 2f_{i-1,j,k}\right)/6, & U_{i,j,k} < 0.
\end{cases} \quad (6.18)$$

Note that the diffusive and convective terms are discretized at the grid vertices $(i, j, k)$. This is a key advantage, in contrast to the staggered grid, that our method inherits from the semi-staggered grid structure. The availability of all three components of velocity avoids the need to interpolate velocity across different grids. This not only simplifies the discretization of the momentum equation, but also avoids the use of the Christoffel symbol. Solving momentum equations at grid vertices also allows for the direct enforcement of boundary conditions, and thereby the use of ghost cells becomes unnecessary. This property is particularly desirable in curvilinear grids. Furthermore the discretization of the diffusive and convective terms at points near the physical boundaries are carried out straightforwardly, without compromising the order of accuracy at places where high accuracy is usually required. From the computational point of view, the linear discrete operator $L(.)$ is identical for all three components of the momentum equations and thus can be calculated (or stored) only once and be used for all three components.
Note that the discretization of the energy equation is similar to the provisional momentum equation, due to the presence of the identical differential operators in both equations, and also the same grid placement of the unknowns, \textit{i.e.} both velocity components and temperature are stored at grid vertices. In the remaining of this paper, we only discuss the discretization of the momentum equation while keeping in mind that the same spatial discretization of the convective and diffusive operators, $\mathcal{N}(\cdot)$ and $\mathcal{L}(\cdot)$, are used for both momentum and energy equations.

6.2.3 Projection step

Before the projection step, we have obtained the intermediate velocity $\tilde{u}$ using equations 6.14-6.15. In the projection step, the intermediate velocity field $\tilde{u}$ is projected to a divergence-free vector space. This leads to a Poisson-Neumann problem to obtain a projection function $\phi$. Our goal is to present a consistent discretization of the Poisson-Neumann equation in which (1) the hydrostatic mode is the only vector in the null space of the discretization matrix, and (2) the discrete continuity is satisfied exactly.

In this section we present the discretization of the Poisson-Neumann problem in the generic form, which is then followed by discretizations on semi-staggered grid and staggered grid arrangements, where two discretizations are compared against each other. The divergence-free velocity field at time level $n+1$ is obtained from:

$$
\frac{u^{n+1} - \tilde{u}}{\Delta t} = G_x(\phi^{n+1}). 
$$

Equation 6.19 is projected to the normal vector of $\xi^i$-constant face by taking the inner product with vector $\beta^i$. This follows:

$$
U^{n+1} = \tilde{U} - \Delta tG^\xi(\phi^{n+1}).
$$

(6.20)
Taking divergence of the equation 6.20 and enforcing $\mathcal{D}(\mathbf{U}^{n+1}) = 0$ yields:

$$
\mathcal{D}\mathcal{G}^{\xi}(\phi^{n+1}) = \frac{\mathcal{D}(\mathbf{U})}{\Delta t},
$$

(6.21)

$$
\mathcal{G}^{\xi}(\phi^{n+1}) \cdot \mathbf{n}\big|_{\Gamma} = 0.
$$

(6.22)

The generic operators $\mathcal{D}$, $\mathcal{G}_x$ and $\mathcal{G}^\xi$ are introduced in table 6.1, and $\Gamma$ denotes the boundary of the computational domain. Equation 6.22 is the discrete representation of the homogeneous Neumann boundary condition where $\mathbf{n}$ represents the normal vector to the physical boundary. The condition given by equation 6.22 at an $\xi^1$-constant boundary, for example, effectively becomes $\mathcal{G}^{\xi_1}(\phi^{n+1}) = 0$, since $\mathcal{G}^{\xi_1}(\phi^{n+1})$ is, in fact, the projection of the vector $\mathcal{G}_x(\phi^{n+1})$ to the normal direction of the $\xi^1$-plane. In the next section we discuss two different strategies to obtain discrete operators for equation 6.21.

- Discretization on semi-staggered grid

  We first consider the discretization of equation 6.21 on the semi-staggered grid. In the semi-staggered grid the contra-variant flux vector is only available at cell vertices and the projection function is stored at the cell center. Thus the divergence operator is discretized at the cell center, while making use of contra-variant flux at cell vertices. This follows:

$$
\mathcal{D}_{ssg}(\mathbf{U}) \bigg|_{i+1/2,j+1/2,k+1/2} := \frac{1}{J} \left( \frac{\delta_1(\mathbf{U}^1)^k}{\delta \xi^1} + \frac{\delta_1(\mathbf{U}^2)^k}{\delta \xi^2} + \frac{\delta_1(\mathbf{U}^3)^k}{\delta \xi^3} \right),
$$

(6.23)

where $ssg$ refers to the semi-staggered grid and each overbar represents an average of the discrete gradient operation at adjacent vertices, as introduced in equation 6.12. The the $\xi^1$-component of the gradient operator is discretized at the cell vertices as:

$$
\mathcal{G}^{\xi_1}_{ssg}(\phi) \bigg|_{i,j,k} := B^{11} \frac{\delta_1(\phi)^k}{\delta \xi^1} + B^{12} \frac{\delta_1(\phi)^k}{\delta \xi^2} + B^{13} \frac{\delta_1(\phi)^k}{\delta \xi^3}.
$$

(6.24)
Figure 6.2: Pressure-Poisson discretization on uniform Cartesian grid for (a) semi-staggered grid, (b) staggered grid

$G_{ssg}^{\xi 2}(\phi)$ and $G_{ssg}^{\xi 3}(\phi)$ are calculated similarly. Both divergence and gradient approximations given by equations 6.23 and 6.24 are second-order accurate. The resulting discretization coefficients for a general two-dimensional curvilinear grid is given in table 6.2. Note that the mesh skewness tensor $B_{ij}$ is symmetric and $B_{ij}$ with $i > j$ is replaced with $B_{ji}$ for conciseness. Specifically, for a uniform Cartesian grid, with $\Delta x_1$ and $\Delta x_2$ denoting the grid spacings in the $x_1$ and $x_2$ directions respectively, the stencil coefficients are shown in figure 6.2(a). It is evident that the discretization stencil annihilates for two spurious modes as indicated by several authors [69, 27]. These spurious modes appearing in the discretization of $D_{ssg}G_{ssg}^{\xi}$ are not desirable and in the next section we adapt a discretization of $DG^{\xi}$, in which the spurious modes are absent.

- Discretization on staggered grid

Discretization of the Poisson-Neumann problem given by equations 6.21-6.22 on staggered grid ensures the absence of any spurious mode while satisfying the discrete continuity exactly. We, thus, require to interpolate the contra-variant flux vector from the cell vertices to cell face centers. The interpolation of $U^i$ component is carried out in $\xi^i$-constant plane. The interpolation has to be at least third-order accurate. The interpolation with second-
order accuracy, the same as the order of accuracy of the method, leads to non-vanishing errors in steady-state simulations and non-satisfactory results in time-dependent solutions. The discrete divergence operator defined on a staggered grid is given by:

\[ \mathcal{D}_{sg}(U) \bigg|_{i+1/2,j+1/2,k+1/2} := \frac{1}{J} \left( \frac{\delta_1(U^1)}{\delta \xi^1} + \frac{\delta_1(U^2)}{\delta \xi^2} + \frac{\delta_1(U^3)}{\delta \xi^3} \right). \]  

(6.25)

Note that the divergence on staggered grid acts directly on the interpolated contra-variant flux vector to cell faces. The \( \xi^1 \)-component of the gradient operator on staggered grid evaluated at grid \( \xi^1 \)-constant face reads as:

\[ \mathcal{G}^{\xi^1}_{sg}(\phi) \bigg|_{i,j+1/2,k+1/2} := B^{11} \frac{\delta_1(\phi)}{\delta \xi^1} + B^{12} \frac{\delta_1(\phi)}{\delta \xi^2} + B^{13} \frac{\delta_1(\phi)}{\delta \xi^3}. \]  

(6.26)

To evaluate the cross-derivative in equation 6.26, some sort of averaging should be incorporated. The discretization coefficients of the discrete operator of the Poisson-Neumann problem on a two-dimensional staggered grid \( \mathcal{D}_{sg} \mathcal{G}^{\xi}_{sg} \) are shown in table 6.2. The stencil coefficients for a uniform Cartesian grid are also shown in figure 6.2(b). A comparison between the semi-staggered and staggered grid discretization of the Poisson-Neumann prob-
lem given by equations 6.21-6.22 reveals that:

1. Discretization of the pressure-Poisson equation on a semi-staggered grid is liable to spurious eigen-modes, whereas for the discrete Poisson-Neumann operator on staggered grid, as employed here, the spurious-modes are eliminated.

2. Discretization of $DG^\xi$ on staggered grid, results in a diagonally dominant matrix. However the diagonal dominance of the coefficient matrix on semi-staggered grid is not guaranteed. In the case of discretization of $D_{ssg}G_{ssg}^\xi$ on uniform Cartesian grid, the diagonal dominance is only retained if $\Delta x_1 = \Delta x_2$ (see figure 6.2(a)).

3. Neither of the discretizations requires “numerical” or ad hoc boundary conditions for the projection function $\phi^{n+1}$ to be invoked. It can be easily verified that the Neumann boundary condition given by equation 6.22 is implicitly enforced by zeroing out the stencil coefficients evaluated at the physical boundary when the discretization of the operator $DG^\xi(\phi^{n+1})$ is carried out at boundary cells. This favorable feature is in accordance with the continuous system of Navier-Stokes equations, where no physical boundary condition is provided for pressure, and is shared by any grid layout where pressure is stored at the cell center.

In the current method, we employ the staggered grid discretization of the Poisson-Neumann problem due to its advantages to the semi-staggered grid discretization. This is where our algorithm departs from the semi-staggered method.

### 6.2.4 Summary of the algorithm

Here we present a complete summary of the current numerical algorithm to solve the time-dependent incompressible Navier-Stokes equations on curvilinear grids in a single time step:

1. Solve equations 6.14-6.15 to obtain the intermediate velocity vector $\tilde{u}$ at cell vertices.

2. Calculate contra-variant flux vector $\tilde{U}$ at cell vertices using equation 6.7.
3. Interpolate (at least a third order interpolation) $\tilde{U}$ from cell vertices to cell face centers (figure 6.3(a)).

4. Solve the Poisson-Neumann problem 6.21-6.22 using divergence operator given by equation 6.25 and gradient operator given by equation 6.26 which are constructed on the staggered grid (figure 6.3(b)).

5. Update the contra-variant flux vector at cell face centers using equation 6.20 to obtain the divergence-free vector field $U^{n+1}$ at cell face centers (figure 6.3(c)).

6. Interpolate (at least a third order interpolation) $U^{n+1}$ from face cell centers to cell vertices (figure 6.3(d)).

7. Calculate Cartesian velocity components $u^{n+1}$ from contra-variant flux vectors $U^{n+1}$ at cell vertices using equation 6.9.

8. Solve the energy equation to obtain temperature $\theta^{n+1}$.

6.3 Results

6.3.1 Taylor-Green vortex

The Taylor-Green vortex problem which presents an exact two dimensional solution to the unsteady incompressible Navier-Stokes equations is used as the first problem to validate the proposed numerical procedure. The exact solution for the domain $0 \leq x_1, x_2 \leq \pi$ is given by:

$$u_1^e(x_1, x_2, t) = -\cos x_1 \sin x_1 e^{-2t},$$

$$u_2^e(x_1, x_2, t) = \sin x_1 \cos x_2 e^{-2t},$$

$$p^e(x_1, x_2, t) = -\frac{1}{4}(\cos 2x_1 + \cos 2x_2) e^{-4t}.$$

To test the numerical method a highly curvilinear grid is used which includes highly skewed grid cells in the domain and it is also non-orthogonal at the boundaries. Such a grid is
Figure 6.3: Schematic of the projection step algorithm (a) interpolating $\tilde{U}^i$ to its corresponding face center; (b) constructing $DG^2$ and solving pressure-Poisson problem; (c) updating the intermediate velocity $\tilde{U}$ to divergence-free velocity field $U^{n+1}$ located at the face center; (d) interpolating $U^{n+1}$ from cell face center to cell vertex.

generated by

\begin{equation}
x_{1,i,j} = L \frac{i - 1}{im - 1} \left\{ \frac{a(im - i)}{(im - 1)} \sin \left( \omega \frac{j - 1}{jm - 1} H \right) + 1 \right\} \quad i = 1, \ldots, im \quad j = 1, \ldots, jm,
\end{equation}

\begin{equation}
x_{2,i,j} = H \frac{j - 1}{jm - 1} \left\{ \frac{a(jm - j)}{(jm - 1)} \sin \left( \omega \frac{i - 1}{im - 1} L \right) + 1 \right\} \quad i = 1, \ldots, im \quad j = 1, \ldots, jm
\end{equation}

where $L$ is the length and $H$ is the height of the rectangular domain and $a$ and $\omega$ tune the amplitude and frequency of the grid lines respectively. Number of grid points in the horizontal and vertical directions are denoted by $im$ and $jm$ respectively. Figure 6.4 shows...
a $65 \times 65$ curvilinear grid with $a = 0.1$ and $\omega = 10\pi$. Boundary condition for the velocity is obtained by enforcing the exact values from the known solution at each time step. For the projection function $\phi$, homogeneous Neumann boundary condition is used. In order to demonstrate the spatial convergence of the method, numerical simulations have been performed on 5 different grids where $33 \times 33$ is the coarsest grid and $161 \times 161$ is the finest. In all simulations, in order to keep the Courant number constant, $\Delta t$ decreases proportionally as grid spacing decreases. The spatial error is calculated at $t = 0.36$ for all different grids. Error is calculated in three different discrete $l_p$-norms, i.e. $p = 1$, $p = 2$ and $p = \infty$. For $p = 1$, $p = 2$ the error is defined as in the following:

$$
\epsilon_p^h = \frac{\|u - u^e\|_p}{N},
$$

(6.29)

where $\|\cdot\|_p$ denotes the discrete $l_p$-norm and $N$ is the total number of points. For $p = \infty$, the error is:

$$
\epsilon_\infty^h = \max(|u - u^e|).
$$

(6.30)
In figures 6.5(a) and 6.5(b), the contours of horizontal velocity component \( u_1 \), obtained from the numerical simulation on two different grid sizes, are compared against the exact solution. The oscillatory behavior in the discrete solution is a consequence of using the sinusoidal grid whose metric terms, also sinusoidal in nature, appear in the truncation error. The discrete solution, however, rapidly converges to the exact solution as a finer mesh with twice as many partitions in each direction is used (see figure 6.5(b)). It is evident that the solution on the finer grid matches the exact solution more precisely even on the highly curvilinear grid. In figure 6.5(c), the convergence of error in \( l_1 \), \( l_2 \) and \( l_\infty \) norms is shown. All error norms converge with at least second order accuracy. Most notably, second-order convergence in \( l_\infty \) norm confirms that the discrete solution converges to the exact solution with at least second-order accuracy at every point in the domain.

6.3.2 Mixed convection in a lid-Driven cavity

Two-dimensional flow in driven cavity has been extensively used for benchmarking of numerical methods. In this section we first present the results for velocity and pressure for a flow in a two-dimensional driven cavity at \( Re = 100 \) and \( Gr = 0 \), which represents the case where the momentum equation is decoupled from the energy equation. In order to test the numerical method, we use an intentionally distorted grid generated using equations 6.27-6.28 with \( L = H = 1 \). We carried out five simulations with grid sizes of \( 33^2 \), \( 49^2 \), \( 65^2 \), \( 97^2 \) and \( 193^2 \). For error analysis the solution on the finest grid is considered as the exact solution [56]. In order to avoid interpolating the solution from coarse grids to the finest grid for error calculation, first the finest grid is generated and the other four remaining grids are created by skipping 1, 2, 3, and 4 points from the finest grid.

In figure 6.6(a), the horizontal velocity component at the vertical mid-plane is compared with that carried out by Ghia et al.[32], in which a uniform Cartesian grid was used. Nevertheless a good agreement is obtained considering that results plotted in figure 6.7 are obtained using a grid that is highly curvilinear and is twice coarser than that used by Ghia et al. In figure 6.6(b), the contours of projection function \( \phi \) are shown. The contour lines
Figure 6.5: (a) and (b) Horizontal velocity contours. Dashed: hybrid semi-staggered grid (HSSG); solid: exact solution; (a) 33 × 33 grid points; (b) 65 × 65 grid points; (c) spatial convergence for Taylor-Green vortex problem.

are in the range of (−0.5, 0.5) with equal increments of 0.05. Due to the construction of the divergence and gradient operators on the staggered grid layout, no spurious pressure behavior is observed. High values of vorticity at the two top corners leads to the progressive concentration of contour lines in these regions. The convergence of the solution of the
driven cavity flow in $l_p$-norms for $p = 1$, $p = 2$ and $p = \infty$ is presented in Table 6.3. Both $l_1$- and $l_2$-norms show at least second order convergence, however $l_\infty$-norm shows poorer convergence. This is due to the discontinuity of the value of the horizontal velocity component at the two top corners of the cavity. This causes the derivative of the velocity at these corners to become unbounded and therefore strong convergence of the numerical solution at these two corners cannot be obtained.

To validate the proposed numerical method for solving energy equation on curvilinear grids, we solve mixed convection on a square lid-driven cavity with non-dimensional length of one, i.e. $\Omega = [0, 1] \times [0, 1]$. The top wall is moving at the constant non-dimensional velocity of $u_1 = 1$ and its temperature is kept at the non-dimensional value of $\theta = 1$. The temperature of the bottom wall is set to be at $\theta = 0$. Adiabatic boundary condition is used
Figure 6.7: Lid-driven cavity: (a) and (b) $Re = 400$ and $Gr = 10^4$; (a) $\theta$ at $x_1 = 0.5$; (b) Temperature filled contours; (c) and (d) $Re = 400$ and $Gr = 10^6$; (c) $\theta$ at $x_1 = 0.5$; (d) Temperature filled contours.

for two side wall. Two cases have been considered with Grashof numbers $Gr = 10^4$ and $Gr = 10^6$. The Reynolds and Prandtl numbers in both cases are $Re = 400$ and $Pr = 0.71$. This problem specification is identical to that used by Iwatsu et al. [46]. A curvilinear grid is generated using equations 6.27-6.28 with $a = 0.1$ and $\omega = 10\pi$. The grid size is $121 \times 121$, a finer grid compared to the case with $Re = 100$, since the current cases have a higher Reynolds number of $Re = 400$ which necessitates a more refined resolution. The numerical solution remained virtually unchanged with further refinement of the grid.
The temperature profiles at the mid vertical plane are compared with computational study carried out by Iwatsu et al. [46] in figures 6.7(a) and 6.7(c). In both cases very good agreements are observed keeping in mind that in the present work a highly distorted grid is used. Temperature contours for both cases with $Gr = 10^4$ and $Gr = 10^6$ are shown in figures 6.7(b) and 6.7(d) respectively. For $Gr = 10^4$, the ratio of inertial forces to buoyancy forces, i.e. $Gr/Re^2 = 0.0625$, is small and the flow inside the cavity is well-mixed. For the higher value of $Gr = 10^6$, buoyancy forces become dominant and the flow in the middle and the bottom of the cavity is nearly stagnant, resulting in horizontal isothermal lines. In the upper part of the cavity, the convection induced by the lid shear movement results in mixing in that region.

6.3.3 Flow past cylinder

In this case, we solve the time-dependent incompressible flow past a circular cylinder in the low and moderate Reynolds number regimes. Reynolds number is defined as $Re = \frac{U_\infty D}{\nu}$ where $U_\infty$ is the upstream velocity and $D$ is the cylinder diameter. For $Re \leq 46$ flow remains two-dimensional, steady-state and symmetric about the center line of the wake [42]. For $Re > 46$, the steady-state flow becomes unstable and the vortex-shedding begins to appear in the flow, giving rise to a time-dependent flow. As the Reynolds number increases the self-sustained time-dependent flow remains two-dimensional, while the amplitude of the oscillations increases. This behavior persists until $Re = 188.5$ where the flow becomes absolutely unstable and three-dimensional disturbances begin to grow. Simulations for $Re = 40, 50, 100$ and $150$ were carried out using a multi-block curvilinear grid around the cylinder as shown in figure 6.8. The computational domain extends $20D$ upstream and $30D$ downstream of the cylinder. In the normal direction to the free stream flow, a $20D$ extension is considered on either side of the cylinder. A total of 290 and 100 grid points are used in the $x_1$ and $x_2$ direction respectively. A total number of 161 points are circumferentially distributed around the cylinder. Throughout the simulations a constant $CFL = 0.6$ is used and $\Delta t$ is calculated accordingly. Table 6.4 shows the comparison of the
Figure 6.8: Multi-block H-grid generated around the cylinder. A total of 161 points are uniformly distributed around the cylinder and a total of 290 and 100 points are used in the $x_1$ and $x_2$ directions.

Table 6.4: Comparison of Strouhal number and drag coefficient with measurements

<table>
<thead>
<tr>
<th>$Re$</th>
<th>$C_D$</th>
<th>$C_D$ Henderson ([42])</th>
<th>St</th>
<th>St (Williamson [90])</th>
</tr>
</thead>
<tbody>
<tr>
<td>40</td>
<td>1.544</td>
<td>1.544</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>50</td>
<td>1.445</td>
<td>1.449</td>
<td>0.121</td>
<td>0.123</td>
</tr>
<tr>
<td>100</td>
<td>1.344</td>
<td>1.350</td>
<td>0.160</td>
<td>0.164</td>
</tr>
<tr>
<td>150</td>
<td>1.331</td>
<td>1.333</td>
<td>0.178</td>
<td>0.183</td>
</tr>
</tbody>
</table>

Strouhal number and the drag coefficient of the results obtained from the current numerical algorithm with two representative experimental data. Strouhal number is not provided for $Re = 40$ where the flow remains steady-state. In all cases a very good agreement is observed. Figures 6.9(a) and 6.9(b) show the drag and lift coefficients at $Re = 150$ for the first 120 non-dimensional time units. In the current simulation, the flow is initialized with the uniform velocity everywhere in the domain which results in zero lift at the beginning of the simulation (see figure 6.9(b)). However the von Karman instability is generated without using numerical perturbations. The flow undergoes a transition and the alternating vor-
tex appears in the cylinder wake leading to a dominantly single-frequency flow behind the cylinder. The flow then becomes stable after \( t = 80 \) and remains statistically steady-state thereafter (see figure 6.9(b)). In figure 6.9(c) the time-averaged pressure and streamlines are shown for \( Re = 100 \). This figure shows the development of the adverse pressure gradient that leads to the flow separation behind the cylinder. All pressure and streamline contours are smooth reflecting statistical and numerical convergence. Figure 6.9(d) shows the comparison of the time-averaged separation angle with those in the existing numerical/experimental studies in the literature. Note that there are 161 grid points distributed around the cylinder and \( \Delta \theta \) between two consecutive grid points on the perimeter of the cylinder is approximately two degrees. Nevertheless a good agreement is observed over a range of Reynolds number and four different published studies.

6.4 Concluding remarks

We have proposed a hybrid staggered/semi-staggered numerical procedure for solving the time-dependent incompressible Navier-Stokes equations on general curvilinear grids. In our method, all velocity components are located at the cell vertices and pressure at the cell center, similar to the semi-staggered grid layout. We adopt a staggered grid approach, however, to solve the Poisson-Neumann equation for the pressure or the corresponding projection function. This advantageous combination results in several features that are particularly favorable for solving the incompressible Navier-Stokes on general curvilinear grids. These features are summarized below:

1. The discretization of the convective and diffusive differential operators are facilitated by discretizing all momentum components at the same location.

2. Velocity boundary conditions are directly enforced at grid vertices on the boundary and the use of ghost layer is avoided.

3. The solution of Poisson-Neumann problem is free of any spurious modes.

4. The resulting matrix discretization of Poisson-Neumann problem is diagonally dom-
Figure 6.9: Flow over cylinder: (a) drag coefficient ($C_d$) at $Re = 150$; (b) lift coefficient ($C_l$) at $Re = 150$; (c) time-averaged pressure and streamlines for $Re = 100$; (d) mean separation angle $\theta$ for flow over cylinder with $\theta$ measured from the stagnation point.
inant even on highly curvilinear grids.

5. Since pressure is stored at cell center, no *ad hoc* pressure boundary condition is required.

6. Discrete continuity is satisfied exactly.

7. Cartesian velocity is chosen as the dependent variable, and the Christoffel symbol does not appear in the transformed equations.

We have successfully tested the current numerical algorithm for Taylor-Green vortex problem, flow in driven cavity and flow over cylinder, which are well-documented benchmarks. In all cases very good agreements are obtained with the published analytical, experimental or numerical results which demonstrate the capability of the presented method to simulate incompressible flow on general curvilinear grids.
Chapter 7
A Symmetric discretization for Pressure-Poisson Equation

7.1 Introduction

In this work we present a symmetric finite difference discretization of the Poisson-Neumann problem on curvilinear grids. The solution to the Poisson-Neumann problem is not unique and is determined up to an additive constant. The solvability condition for this problem is guaranteed if the coefficient matrix of the linear system is symmetric. However care must be taken when discretizing the Poisson-Neumann problem on curvilinear grids, due to the appearance of the cross-derivative terms in the transformed equation which can simply result in a non-symmetric discretization. We propose a symmetric discretization that guarantees that the solvability condition is met while achieving better accuracy. Two validation problems are presented. A two dimensional problem on two highly curvilinear grids is solved. Second order accuracy in $l_2$-norm and first order accuracy in $l_\infty$-norm is demonstrated. In the second problem the applicability of the method is tested in a three dimensional time-dependent incompressible flow in a curved tube. The numerical solution is found to compared well with experimental measurements.

Projection and pressure-correction methods appear in time integration of the incompressible Navier-Stokes equations, in which a Poisson-Neumann problem is solved to project the intermediate velocity field to a divergence-free space. The solution of the Poisson-Neumann problem is determined up to a constant. In the discretized problem, this imposes a solvability condition whose satisfaction can be guaranteed if the coefficient matrix is symmetric. In the current study a symmetric finite difference discretization of the Poisson-Neumann problem is suggested on curvilinear grids.

We consider the incompressible Navier-Stokes equations given by:
\[
\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} = -\frac{\nabla p}{\rho} + \nu \nabla^2 \mathbf{v} \tag{7.1a}
\]
\[
\nabla \cdot \mathbf{v} = 0. \tag{7.1b}
\]

Time advancement of the system of equations given by 7.1 using projection or pressure correction methods requires solving a Poisson equation with homogeneous or inhomogeneous Neumann boundary conditions. Here, for the sake of simplicity, we only consider the homogeneous case. However the extension to inhomogeneous Neumann boundary conditions is straightforward. The Poisson-Neumann problem is given by:

\[
\nabla^2 \psi = \nabla \cdot \tilde{\mathbf{v}} \text{ in } \Omega, \tag{7.2a}
\]
\[
\frac{\partial \psi}{\partial \mathbf{n}} = 0 \text{ on } \Gamma. \tag{7.2b}
\]

Equation 7.2 admits solution up to an additive constant, properly reflecting the same behavior that pressure exhibits in equation 7.1. The finite difference discrete representation of equation 7.2 is expressed as:

\[
\mathbb{J}^{-1} \mathbb{L} \phi = \mathbb{J}^{-1} \mathbf{f}, \tag{7.3}
\]

where

\[
\mathbf{f} = \mathbb{J} \mathbb{D} \mathbf{u}.
\]

Here \( \mathbb{J}^{-1} \mathbb{L}(\sim) \) is the discrete representation of \( \nabla^2(\sim) \) along with its boundary operator \( \frac{\partial(\sim)}{\partial \mathbf{n}} \) and \( \mathbb{J} \) is a diagonal matrix whose diagonal entries are the volume (in 3D)/area (in 2D) of computational cells.

A consistent discretization of equation 7.2 must meet the following conditions: (1) the constant vector \( \mathbf{e} = (1 \ 1 \ 1 \ldots 1)^T \) must lie in the null space of \( \mathbb{L} \). The constant vector \( \mathbf{e} \) is the discrete equivalent of the hydrostatic mode. (2) Irrespective of the discretization of
\( \nabla^2(\sim) \) and \( \frac{\partial (\sim)}{\partial n} \) in equation 7.2, the right hand side vector in \( \mathbb{L}\phi = f \) must be orthogonal to \( \mathbf{e} \). In fact \( \mathbf{e}^T f \) is the discrete equivalent of the global mass conservation, \( i.e.: \)

\[
\int_{\Omega} \nabla \cdot \tilde{\mathbf{v}} \, dx = \oint_{\Gamma} \tilde{\mathbf{v}} \cdot \mathbf{n} \, dx = 0.
\]

This condition is easily met in the discrete representation, where the divergence of the velocity field is approximated as the net volumetric fluxes at cell faces (in 3D)/edges (in 2D). These fluxes “telescope” when they are summed over all cells and only the boundary fluxes remain. The sum of the boundary fluxes, however, must vanish as it represents the discrete global mass conservation. Given that \( \mathbf{e} \) is in the null space of \( \mathbb{L} \) and \( \mathbf{e}^T f = 0 \), the symmetry of \( \mathbb{L} \) becomes a sufficient condition to satisfy the solvability condition [36], as \( \mathbf{e} \) is also in the null space of \( \mathbb{L}^T \).

Existing methods for finite difference/volume discretization of equation 7.2 on transformed curvilinear grids do not preserve the self-adjointness of the differential operator \( \nabla^2(\sim) \) to the discrete operator \( \mathbb{L}(\sim) \). Several researchers have, instead, chosen to modify the right hand side vector \( f \) to ensure the solvability condition for solving equation 7.3 [43, 16, 71]. Evidently, any perturbation to the right hand side of equations 7.3 results in the failure of satisfying discrete discontinuity exactly. This loss of accuracy is particularly unsatisfactory in direct and large eddy simulations of the turbulent flows where the physical accuracy is highly demanded [60, 95].

In the current work, we present a finite difference representation of equation 7.2 on general curvilinear grids in which \( \mathbb{L} \) is symmetric, and \( \mathbf{e} \) is the only vector in the null space of matrix \( \mathbb{L} \), thus automatically satisfying the solvability condition for solving 7.3. We also show that the symmetric discretization is more accurate than existing non-symmetric discretizations.

### 7.2 Equations

In the projection step, the intermediate velocity field \( \tilde{\mathbf{v}} \) is projected to a divergence-free vector space. This leads to a Poisson-Neumann problem to obtain a projection function \( \psi \).
which includes the following two steps:

1. Solving a Poisson-Neumann problem where before spatial discretization takes the form of:

\[
D_x G_x (\psi^{n+1}) = \frac{D_x(\tilde{v})}{\Delta t} \quad \text{in } \Omega, \quad (7.4a)
\]
\[
G_x(\psi^{n+1}) \cdot n = 0 \quad \text{on } \Gamma. \quad (7.4b)
\]

2. Projecting the intermediate vector field \(\tilde{v}\) onto a divergence-free space:

\[
v^{n+1} = \tilde{v} - \Delta tG_x(\psi^{n+1}). \quad (7.5)
\]

We solve equations 7.4 and 7.5 on complex geometries using the non-orthogonal body-fitted grid. The Cartesian coordinates \(\{x_i\}\) are mapped to the curvilinear coordinates by the transformation \(\xi^j = \xi^j(x_i)\) characterized by the Jacobian of the transformation \(J = \left| \frac{\partial x_i}{\partial \xi^j} \right|\). This follows [26]:

\[
D_\xi G_\xi (\psi^{n+1}) = \frac{D_\xi(\tilde{V})}{\Delta t} \quad \text{in } \Omega, \quad (7.6a)
\]
\[
G_\xi(\psi^{n+1}) = 0 \quad \text{on } \xi^i\text{-constant boundary}, \quad (7.6b)
\]

where \(B^{ij}\) is a symmetric second-order tensor given by:

\[
B^{ij} = J \frac{\partial \xi^i}{\partial x_k} \frac{\partial \xi^j}{\partial x_k}, \quad (7.7)
\]
\[
V^j = v \cdot \beta^j. \quad (7.8)
\]

The vector \(\beta^j\) is orthogonal to \(\xi^j\)-constant face and is given by:

\[
(\beta^j)_i = J \frac{\partial \xi^j}{\partial x_i}, \quad (7.9)
\]
Equation 7.5 is projected to the normal vector of $\xi^i$-constant face by taking the inner product of $\beta^i$. This follows:

$$
V^{n+1} = \widetilde{V} - \Delta t G^\xi (\psi^{n+1}).
$$

### 7.3 Discretization

Our goal is to present a consistent finite difference discretization of the Poisson-Neumann problem on general curvilinear grids in which: (1) the resulting coefficient matrix is symmetric; (2) the hydrostatic mode is the only vector in the null space of the discretization matrix; and (3) the discrete continuity is satisfied exactly. The same coordinate transformation as explained in section 6.2 is used here. The computational cell is similar to that of figure 6.1 with slight modifications as shown in figure 7.1. Metric terms are also calculated.
Table 7.1: Continuous and discrete differential operators.

<table>
<thead>
<tr>
<th>Descriptor</th>
<th>Continuous operator</th>
<th>Discrete operator</th>
<th>Matrix operator</th>
</tr>
</thead>
<tbody>
<tr>
<td>co-variant divergence</td>
<td>$D_x(v) := \frac{\partial v_i}{\partial x_i}$</td>
<td>$D_x(u) := \frac{\delta u_i}{\delta x_i}$</td>
<td>-</td>
</tr>
<tr>
<td>co-variant gradient</td>
<td>$G_x(\psi) := \frac{1}{J} \beta^j_i \frac{\partial \psi}{\partial \xi^j}$</td>
<td>$G_x(\phi) := \frac{1}{J} \beta^j_i \frac{\delta \phi}{\delta \xi^j}$</td>
<td>-</td>
</tr>
<tr>
<td>contra-variant divergence</td>
<td>$D^\xi(V) := \frac{1}{J} \frac{\partial V^i}{\partial \xi^i}$</td>
<td>$D^\xi(U) := \frac{1}{J} \frac{\delta U^i}{\delta \xi^i}$</td>
<td>$\mathcal{D}^\xi(U)$</td>
</tr>
<tr>
<td>contra-variant gradient</td>
<td>$G^\xi(\psi) := B^{ij} \frac{\partial \psi}{\partial \xi^j}$</td>
<td>$G^\xi(\phi) := B^{ij} \frac{\delta \phi}{\delta \xi^j}$</td>
<td>$\mathcal{G}^\xi(\phi)$</td>
</tr>
</tbody>
</table>

as demonstrated in section 6.2.1.

7.3.1 Discrete operators

The discrete operators as defined by equations 6.12 and 6.13 are also used in this chapter. The differential continuous and discrete operators are given in table 7.1. This table is similar to table 6.1 with some minor modifications in notation.

7.3.2 Discretization of Poisson-Neumann

The discrete representation of the Poisson-Neumann problem in curvilinear grids given by equation 7.11 takes the generic form:

\[ D^\xi G^\xi(\phi^{n+1}) \bigg|_{i+\frac{1}{2},j+\frac{1}{2}} = D^\xi(\tilde{U}) \quad \text{for all cells,} \quad (7.11a) \]

\[ G^\xi(\phi^{n+1}) = 0 \quad \text{on } \xi^i-\text{constant boundary,} \quad (7.11b) \]

and the resulting linear system becomes:

\[ \mathbb{L}\phi = f \quad (7.12) \]
where:

\[ L = JD^\xi G^\xi \]
\[ f = JD^\xi \tilde{U}. \]

We intend to show that it is possible to discretize \( D^\xi \) and \( G^\xi \) such that:

\[ (JD^\xi G^\xi)^T = (G^\xi)^T (JD^\xi)^T. \]

From the above relation the symmetry of the coefficient matrix \( L \) in equation 7.12 will directly follow. In the next section we present a symmetric discretization for problem 7.11. We carry out the discretization in two dimensions for the sake of simplicity. We also present the extension to three dimensions in section 7.3.2.

- **Symmetric discretization**

  For clarity we consider equation 7.11 in the expanded form. Thus:

  \[
  D^\xi g^\xi(\phi) = \frac{1}{J} \left\{ \frac{\delta}{\delta \xi^1} (B^{11} \frac{\delta \phi}{\delta \xi^1}) + \frac{\delta}{\delta \xi^1} (B^{12} \frac{\delta \phi}{\delta \xi^2}) + \frac{\delta}{\delta \xi^2} (B^{21} \frac{\delta \phi}{\delta \xi^1}) + \frac{\delta}{\delta \xi^2} (B^{22} \frac{\delta \phi}{\delta \xi^2}) \right\}. \tag{7.13}
  \]

Let \( L_{11}, L_{12}, L_{21} \) and \( L_{22} \) represent the coefficient matrices resulting from the discretization of \( \frac{\delta}{\delta \xi^1} (B^{11} \frac{\delta \phi}{\delta \xi^1}), \frac{\delta}{\delta \xi^1} (B^{12} \frac{\delta \phi}{\delta \xi^2}), \frac{\delta}{\delta \xi^2} (B^{21} \frac{\delta \phi}{\delta \xi^1}) \) and \( \frac{\delta}{\delta \xi^2} (B^{22} \frac{\delta \phi}{\delta \xi^2}) \) respectively. Thus we have:

\[
L = L_{11} + L_{12} + L_{21} + L_{22}. \tag{7.14}
\]

**Discretization of** \( \frac{\delta}{\delta \xi^1} (B^{11} \frac{\delta \phi}{\delta \xi^1}) \) **and** \( \frac{\delta}{\delta \xi^2} (B^{22} \frac{\delta \phi}{\delta \xi^2}) :**

These terms, also present in a Cartesian grid, can be discretized as it was shown by Gibou et al. [33]. This follows:
\[
\frac{\delta}{\delta \xi_1} (B_{11} \frac{\delta \phi}{\delta \xi_1}) \bigg|_{i+1/2,j+1/2} := B_{i+1,j+1/2}^{11} (\phi_{i+3/2,j+1/2} - \phi_{i+1/2,j+1/2}) - B_{i,j+1/2}^{11} (\phi_{i+1/2,j+1/2} - \phi_{i-1/2,j+1/2}) \tag{7.15}
\]

and similarly and more compactly using the discrete operator given by equation 6.13:

\[
\frac{\delta}{\delta \xi_2} (B_{22} \frac{\delta \phi}{\delta \xi_2}) \bigg|_{i+1/2,j+1/2} := \frac{\delta_1}{\delta \xi_2} (B_{22} \frac{\delta \phi}{\delta \xi_2}) \tag{7.16}
\]

It is evident that the above schemes produce symmetric coefficient matrices \(L_{11}\) and \(L_{22}\).

**Discretization of \(\frac{\delta}{\delta \xi_1} (B_{12} \frac{\delta \phi}{\delta \xi_2})\) and \(\frac{\delta}{\delta \xi_2} (B_{21} \frac{\delta \phi}{\delta \xi_1})\):**

Similar to the scheme that was used to discretize \(\frac{\delta}{\delta \xi_1} (B_{11} \frac{\delta \phi}{\delta \xi_1})\) we have:

\[
\frac{\delta}{\delta \xi_1} (B_{12} \frac{\delta \phi}{\delta \xi_2}) \bigg|_{i+1/2,j+1/2} := B_{i+1,j+1/2}^{12} \bigg( \frac{\delta \phi}{\delta \xi_1} \bigg|_{i+1/2,j+1/2} - B_{i,j+1/2}^{12} \bigg( \frac{\delta \phi}{\delta \xi_2} \bigg|_{i,j+1/2} \bigg) \tag{7.17}
\]

and \(B_{12}^{12} \bigg( \frac{\delta \phi}{\delta \xi_2} \bigg|_{i,j+1/2} \bigg)\) or \(B_{12}^{12} \bigg( \frac{\delta \phi}{\delta \xi_2} \bigg|_{i+1,j+1/2} \bigg)\) are evaluated at the center of \(\xi^1\)-edge. Therefore some sort of averaging must be used due to the unavailability of \(\phi\) along the \(\xi^1\)-edge. We use the average at the two edge vertices. This follows:

\[
B_{12}^{12} \bigg( \frac{\delta \phi}{\delta \xi_2} \bigg|_{i,j+1/2} \bigg) := \frac{1}{2} \left\{ B_{i,j+1}^{12} \left( \frac{\delta \phi}{\delta \xi_2} \bigg|_{i-1/2,j+1} + \frac{\delta \phi}{\delta \xi_2} \bigg|_{i+1/2,j+1} \right) / 2 \right\} + B_{i,j}^{12} \left( \frac{\delta \phi}{\delta \xi_2} \bigg|_{i-1/2,j} + \frac{\delta \phi}{\delta \xi_2} \bigg|_{i+1/2,j} \right) / 2 \} \tag{7.18}
\]

Replacing equation 7.18 in equation 7.17 and using the operators given by equations 6.12 and 6.13 result in:

\[
\frac{\delta}{\delta \xi_1} (B_{12}^{12} \frac{\delta \phi}{\delta \xi_2}) \bigg|_{i+1/2,j+1/2} := \frac{\delta_1}{\delta \xi_1} (B_{12}^{12} \frac{\delta_1 \phi}{\delta \xi_2}) \tag{7.19}
\]
and similarly:

\[
\frac{\delta}{\delta \xi^2} \left( B_{21} \frac{\delta \phi}{\delta \xi^1} \right) \bigg|_{i+1/2, j+1/2} := \frac{\delta_1}{\delta \xi^2} \left( B_{21} \frac{\delta_1 (\phi)^j}{\delta \xi^1} \right). \tag{7.20}
\]

The resulting coefficient matrices for the cross derivative terms are non-symmetric, \textit{i.e.}:

\[
\mathbb{L}_{12} \neq \mathbb{L}_{12}^T \quad \text{and} \quad \mathbb{L}_{21} \neq \mathbb{L}_{21}^T.
\]

However after taking into account that \( B_{ij} \) is a symmetric tensor, it is easy to verify that:

\[
\mathbb{L}_{12} - \mathbb{L}_{12}^T = \mathbb{L}_{21}^T - \mathbb{L}_{21}. \tag{7.21}
\]

Using equations 7.14 and 7.21 and knowing that \( \mathbb{L}_{11} \) and \( \mathbb{L}_{22} \) are symmetric matrices confirm that \( \mathbb{L} \) is symmetric.

Note that since \( \frac{\delta_1 \phi}{\delta \xi^1} \) and \( \frac{\delta_1 \phi}{\delta \xi^2} \) annihilates for a constant \( \phi \), the hydrostatic mode lies in the null space of \( \mathbb{L} \), \textit{i.e.}:

\[
\mathbb{L} \mathbf{e} = \mathbf{0}.
\]

Besides, matrix \( \mathbb{L} \) can be simply decomposed to:

\[
\mathbb{L} \equiv \mathbb{J} \mathbb{D}^\xi \mathbb{G}^\xi, \tag{7.22}
\]

where \( \mathbb{D}^\xi \) and \( \mathbb{G}^\xi \) are the coefficient matrices of the following discretization schemes:

\[
\mathbb{D}(\mathbf{U}) \bigg|_{i+1/2, j+1/2} := \frac{1}{J} \left( \frac{\delta_1 (U^1)}{\delta \xi^1} + \frac{\delta_1 (U^2)}{\delta \xi^2} \right), \tag{7.23}
\]

\[
\mathbb{G}^\xi (\phi) = \left( \mathbb{G}^{\xi_1} (\phi), \mathbb{G}^{\xi_2} (\phi) \right)^T, \tag{7.24}
\]
Figure 7.2: Schematic of an $\xi^1$-constant boundary with $i = 1$.

where:

$$G^{\xi^1} (\phi) \bigg|_{i,j+1/2} := B^{11} \frac{\delta_1 (\phi)}{\delta \xi^1} + B^{12} \frac{\delta_1 (\phi)}{\delta \xi^2} ,$$

$$G^{\xi^2} (\phi) \bigg|_{i+1/2,j} := B^{22} \frac{\delta_1 (\phi)}{\delta \xi^2} + B^{21} \frac{\delta_1 (\phi)}{\delta \xi^1} .$$

The decomposition of the Laplacian operator to the divergence of the gradient operator as given by equation 7.22 is crucial to ensure that the discrete continuity is satisfied exactly.

- **Neumann boundary condition**

Another interesting property of the symmetric discretization presented in the previous section is the “natural” inclusion of the Neumann boundary condition to the discrete system. It is natural in the sense that, as we show in this section, the discretization scheme that is used for the cells adjacent to a boundary is identical to that of the interior cells with the distinction that all the stencil terms at the physical boundary are zeroed out. This is similar to natural inclusion of boundary condition on Cartesian grids as it is demonstrated by Kim et al. [52].

Let’s consider an $\xi^1$-constant edge, where $i = 1$, as shown in figure 7.2. In this section, we keep $\Delta \xi^1$ and $\Delta \xi^2$ in the discretization for the sake of clarity in the error analysis. Using the symmetric discretization presented in the previous section for equation 7.11 and
zeroing out all $B^{ij}$ terms at edge $i=1$ result in:

\[
\frac{1}{\Delta \xi_1} \left\{ B^{11} \frac{\delta \phi}{\delta \xi_1} \bigg|_{2,j+1/2} + B^{12} \frac{\delta \phi}{\delta \xi_2} \bigg|_{2,j+1/2} \right\} + \frac{1}{2\Delta \xi_2} \left\{ B^{21} \frac{\delta \phi}{\delta \xi_1} \bigg|_{2,j+1} - B^{21} \frac{\delta \phi}{\delta \xi_1} \bigg|_{2,j} \right\} + \frac{1}{\Delta \xi_2} \left\{ B^{22} \frac{\delta \phi}{\delta \xi_2} \bigg|_{3/2,j+1} - B^{22} \frac{\delta \phi}{\delta \xi_2} \bigg|_{3/2,j} \right\} = \frac{U^1_{2,j+1/2} - U^1_{1,j+1/2}}{\Delta \xi_1} + \frac{U^2_{3/2,j+1} - U^2_{3/2,j}}{\Delta \xi_2},
\]

(7.27)

multiplying both sides by $\Delta \xi_1$ and after re-arranging:

\[
B^{11} \frac{\delta \phi}{\delta \xi_1} \bigg|_{2,j+1/2} + B^{12} \frac{\delta \phi}{\delta \xi_2} \bigg|_{2,j+1/2} = \frac{\Delta \xi_1}{\Delta \xi_2} (g_1^1 - g_2^1) + \frac{\Delta \xi_1}{\Delta \xi_2} (g_2^2 - g_2^2) + \frac{\Delta \xi_1}{\Delta \xi_2} (g_3^3 - g_3^3) + (g_2^3 - g_2^3) + (g_3^3 - g_3^3),
\]

(7.28)

where:

\[
\begin{align*}
g^1 &:= -\frac{1}{2} B^{21} \frac{\delta \phi}{\delta \xi_1} \\
g^2 &:= - B^{22} \frac{\delta \phi}{\delta \xi_2} \\
g^3 &:= U^1 \\
g^4 &:= U^2
\end{align*}
\]

Assuming that $O(\Delta \xi_1) \sim O(\Delta \xi_2) \sim O(h)$ and knowing that for an $\xi^1$-constant boundary:

\[
\frac{\delta \phi}{\delta n} = B^{11} \frac{\delta \phi}{\delta \xi_1} + B^{12} \frac{\delta \phi}{\delta \xi_2},
\]
where \( n \) is normal to the boundary \( i = 1 \). Then in equation 7.28 for small \( h \) we have:

\[
B^{11} \frac{\delta \phi}{\delta \xi^1}_{2,j+1/2} + B^{12} \frac{\delta \phi}{\delta \xi^2}_{2,j+1/2} \approx \frac{\delta \phi}{\delta n}_{1,j+1/2} + \mathcal{O}(h)
\]

\[
g^{1}_{2,j+1} - g^{1}_{2,j} \approx \mathcal{O}(h)
\]

\[
g^{2}_{3/2,j+1} - g^{2}_{3/2,j} \approx \mathcal{O}(h)
\]

\[
g^{3}_{2,j+1/2} - g^{3}_{1,j+1/2} \approx \mathcal{O}(h)
\]

\[
g^{4}_{3/2,j+1} - g^{4}_{3/2,j} \approx \mathcal{O}(h)
\]

\[
g^{4}_{3/2,j+1} - g^{4}_{3/2,j} \approx \mathcal{O}(h)
\]

\[
\mathcal{O}(\Delta \xi^1) \approx \mathcal{O}(1).
\]

Using the above relationships, equation 7.27 becomes:

\[
\frac{\delta \phi}{\delta n} \approx \mathcal{O}(h),
\]

which is a first order approximation for the Neumann boundary condition given in equation 7.11. The natural inclusion of the Neumann boundary condition, as explained above, has the following properties:

1. Coefficient matrix \( L \) remains symmetric. This is achieved since the stencil coefficients of the cells adjacent to the boundary that reach to the other interior cells remain intact.

2. The summation of the stencil coefficients for the cell adjacent to the boundary vanishes, which ensures that the hydrostatic mode remains in the null space of \( L \).

In summary \( L \) is a discrete representation of the differential operators in equation 7.11.

- **Extension to three dimensions**

Symmetric discretization of the Poisson-Neumann problem in three dimensions is similar to two dimensional discretization as explained in the previous section. In fact the
discretization of \( \frac{\delta}{\delta \xi_1}(B^{11} \frac{\delta \phi}{\delta \xi_1}), \frac{\delta}{\delta \xi_1}(B^{12} \frac{\delta \phi}{\delta \xi_2}), \frac{\delta}{\delta \xi_2}(B^{21} \frac{\delta \phi}{\delta \xi_1}) \) and \( \frac{\delta}{\delta \xi_2}(B^{22} \frac{\delta \phi}{\delta \xi_2}) \), for instance, is only shifted to the \( \xi^1 - \xi^2 \) plane of \( \xi^3 = k + 1/2 \). In summary the discrete divergence and gradient operator in three dimensions are:

\[
D(U) \bigg|_{i+1/2,j+1/2,k+1/2} := \frac{1}{J} \left( \frac{\delta_1(U^1)}{\delta \xi_1} + \frac{\delta_1(U^2)}{\delta \xi_2} + \frac{\delta_1(U^3)}{\delta \xi_3} \right), \tag{7.29}
\]

\[
\mathcal{G}^\xi(\phi) = \left( \mathcal{G}^{\xi_1}(\phi), \mathcal{G}^{\xi_2}(\phi), \mathcal{G}^{\xi_3}(\phi) \right)^T, \tag{7.30}
\]

where:

\[
\mathcal{G}^{\xi_1}(\phi) \bigg|_{i,j+1/2,k+1/2} := B^{11} \frac{\delta_1(\phi)}{\delta \xi_1} + B^{12} \frac{\delta_1(\phi)}{\delta \xi_2} + B^{13} \frac{\delta_1(\phi)}{\delta \xi_3}, \tag{7.31}
\]

\[
\mathcal{G}^{\xi_2}(\phi) \bigg|_{i+1/2,j,k+1/2} := B^{22} \frac{\delta_1(\phi)}{\delta \xi_2} + B^{21} \frac{\delta_1(\phi)}{\delta \xi_1} + B^{23} \frac{\delta_1(\phi)}{\delta \xi_3}, \tag{7.32}
\]

\[
\mathcal{G}^{\xi_3}(\phi) \bigg|_{i+1/2,j+1/2,k} := B^{33} \frac{\delta_1(\phi)}{\delta \xi_3} + B^{31} \frac{\delta_1(\phi)}{\delta \xi_1} + B^{32} \frac{\delta_1(\phi)}{\delta \xi_2}. \tag{7.33}
\]

### 7.4 Results

#### 7.4.1 Two-dimensional test case

This problem is designed to demonstrate the order of accuracy of the proposed discretization of the Poisson-Neumann problem and compare and contrast its performance on highly curvilinear grids with existing non-symmetric discretizations [94]. We consider the following problem:

\[
\nabla^2 \psi = \nabla \cdot \mathbf{v} \quad \text{in } \Omega, \tag{7.34a}
\]

\[
\frac{\partial \psi}{\partial n} = 0 \quad \text{on } \Gamma, \tag{7.34b}
\]

where \( \Omega = [0, 1] \times [0, 1] \) and

\[
\mathbf{v} = -k \sin(kx_1) \cos(kx_2) \mathbf{e}_{x_1} - k \cos(kx_1) \sin(kx_2) \mathbf{e}_{x_2},
\]
Figure 7.3: Test grids for discretization of Poisson-Neumann problem (a) G1: highly curvi-linear grid \((41 \times 41)\) (b) G2: rough grid \((41 \times 41)\).

and \(k = 2\pi\). The exact solution is:

\[
\psi(x_1, x_2) = \cos(kx_1) \cos(kx_2).
\]  

(7.35)

We solve problem 7.34 on two deliberately distorted grids, as shown in figure 7.3. Grid G1 (figure 7.3(a)) is smooth but it has cells with high skewness. Grid G2 (figure 7.3(b)), on the other hand, is a highly distorted and non-smooth grid. The surface plot of each component of the \(B^{ij}\) tensor is shown in figure 7.4. The dramatic variation of \(B^{ij}\) poses a challenging case for the discrete problem.

We solved equation 7.34 with the symmetric and non-symmetric \([94]\) discretizations. Note that the exact solution as given by equation 7.35 is smooth. The contour plots of the numerical solutions for symmetric and non-symmetric discretizations on both G1 and G2 are shown in figure 7.5.

Clearly the symmetric discretization is more accurate on both grids. Particularly on G2, in the regions with non-smooth change in the slope of grid lines or a sharp change in
grid spacing, symmetric discretization shows a smooth solution. To measure the order of accuracy of the method, we solved problem 7.34 on G1 with five different grid sizes in the range of 21 × 21 to 101 × 101. The discrete $l_2$-norm and $l_{\infty}$-norm for both discretizations is given in table 7.2.

It shows that both methods are second-order accurate in $l_2$-norm and first-order accurate in $l_{\infty}$-norm. However the magnitude of the error of the symmetric discretization is almost one order lower in both norms than that of the non-symmetric discretization. For instance the solution of the symmetric discretization on the 61 × 61 grid is more accurate
in both norms than that of the non-symmetric case on the 101 × 101 grid.

To show the convergence on grid G2, we carried out five simulations on a batch of grids in the range of 51 × 51 to 301 × 301. The error convergence is shown in figure 7.6 and first order convergence in $l_\infty$-norm and second order convergence in $l_2$-norm is achieved. The non-symmetric discretization also has the same order of accuracy, however the value of the error in the symmetric discretization remains lower for all grid sizes.
Table 7.2: Grid convergence study for symmetric and non-symmetric discretizations of Poisson-Neumann problem on G1.

| Grid Size | Symmetric | | | | | | Non-symmetric | | | | | |
|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|
|           | $l_\infty$ | $l_2$ | Order | $l_\infty$ | $l_2$ | Order | $l_\infty$ | $l_2$ | Order |
| 21 x 21   | 1.31 x 10^{-1} | - | 3.25 x 10^{-1} | - | 4.40 x 10^{-1} | - | 1.11 x 10^{-2} | - |       |
| 41 x 41   | 8.46 x 10^{-2} | 0.65 | 1.00 x 10^{-3} | 1.76 | 2.50 x 10^{-1} | 0.84 | 2.94 x 10^{-3} | 1.98 |
| 61 x 61   | 5.53 x 10^{-2} | 1.07 | 4.30 x 10^{-4} | 2.12 | 1.73 x 10^{-1} | 0.93 | 1.32 x 10^{-3} | 2.01 |
| 81 x 81   | 4.01 x 10^{-2} | 1.13 | 2.33 x 10^{-4} | 2.16 | 1.31 x 10^{-1} | 0.98 | 7.47 x 10^{-4} | 2.01 |
| 101 x 101 | 3.12 x 10^{-2} | 1.17 | 1.44 x 10^{-4} | 2.18 | 1.06 x 10^{-1} | 0.96 | 4.79 x 10^{-4} | 2.01 |

Figure 7.6: Error convergence for symmetric and non-symmetric discretization on G2.

In figure 7.7, the computed values of the discrete divergence at cell centers are shown for a 21 x 21 grid. The linear system is solved by a direct solver and thus discrete continuity is satisfied to the order of machine accuracy.

7.4.2 Unsteady incompressible flow in a 90-degree curved tube

In the second example, we solve the time-dependent incompressible Navier-Stokes equations in a 90° pipe bend at a moderate Reynolds number. We intend to demonstrate the performance of the proposed discretization when it is used as the projection step in a com-
plex three dimensional flow. We compare the numerical simulation against the experimental measurements that was carried out by Rindt et al. [76].

The two dimensional schematic of the problem geometry is shown in figure 7.8(a). The radius of the pipe is denoted by $R$. The pipe is consisted of three segments: (1) a horizontal entry segment with the length of $6R$, (2) a $90^\circ$ bend with the centerline radius of $6R$ and (3) the vertical exit segment with the length of $6R$. Figure 7.8(b) shows the curvilinear grid used to solve this problem numerically, which has 61 sections along the streamwise direction. Each section has an identical $51 \times 51$ grid as it is shown in figure 7.8(b).

The inlet velocity is assumed to be the Womersly solution of a fully developed pulsatile flow in pipe[31] which is given by:

$$u_1(r, t) = u_b(r) + u_p(r, t),$$
where \( u_b(r) \) is the fully developed velocity profile in the pipe:

\[
    u_b(r) = 2[1 - \left(\frac{r}{R}\right)^2],
\]

and \( u_p(r, t) \) is the pulsatile flow:

\[
    u_p(r, t) = \frac{k}{i\omega}e^{i\omega t}[1 - \frac{J_0\left(\frac{r\omega}{\nu}\right)}{J_0\left(\frac{R\omega}{\nu}\right)}],
\]

where \( J_0 \) is the zero order Bessel function of the first kind, \( \omega = \frac{2\pi}{T} \) where \( T = 12.3(s) \) is the time period of pulsations, \( \nu = \frac{1}{500}(m^2/s) \) and \( k = 0.375 \). The inlet velocity profile corresponds to a pulsatile flow component of \(-300 < Re < 300\) that is superimposed on a steady fully developed flow of \( Re = 500 \). The time variation of the Reynolds number computed from the inlet velocity and the experimental measurements, for one period, is shown in figure 7.9.

We solve the time-dependent incompressible Navier-Stokes equations using fractional
step method. The viscous terms are discretized using second order accurate finite difference discretization. For convective terms, a third order biased upwind scheme [72] is used. In the first step the momentum equation without pressure gradient is advanced in time using first order Euler scheme. The viscous terms are treated implicitly and convective terms are treated explicitly. In the second step the intermediate velocity field is projected to a divergence-free space using the proposed symmetric discretization.

Each pulsation period is advanced in 1000 time steps, i.e. $\Delta t = 0.0123$. We compare the streamwise velocity profile at $t=T/2$ and $t=T$ with experimental measurements [76]. The measurements are carried out at $\theta = 0^\circ$, $\theta = 22.5^\circ$, $\theta = 45^\circ$, $\theta = 67.5^\circ$ and $\theta = 90^\circ$ where $\theta$ is the section angle across the bend (see figure 7.8(a)). The results for $t=T/2$ and $t=T$ are shown in figures 7.10 and 7.11 respectively and as can be seen very good agreements are observed.

In order to demonstrate the satisfaction of the discrete continuity in three dimensions, we carried out two separate simulations for different tolerance levels for the iterative solver of the Poisson-Neumann problem. We calculated the volumetric flow rate of the flow at inlet section and for $\theta = 22.5^\circ$, $\theta = 45^\circ$, $\theta = 67.5^\circ$ and $\theta = 90^\circ$ along the bend. The
Table 7.3: Volumetric flow rate $Q$ at different pipe sections for relative residual tolerance of $\text{tol}= 1 \times 10^{-5}$.

<table>
<thead>
<tr>
<th>section</th>
<th>$Q(t = T/4)$</th>
<th>$Q(t = T/2)$</th>
<th>$Q(t = 3T/4)$</th>
<th>$Q(t = T)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>inlet</td>
<td>1.25966937926702</td>
<td>0.692401765602920</td>
<td>0.311042098160758</td>
<td>0.878309721297001</td>
</tr>
<tr>
<td>$\theta = 0^\circ$</td>
<td>1.2596693809801</td>
<td>0.692401765350104</td>
<td>0.311042100439914</td>
<td>0.878309721526652</td>
</tr>
<tr>
<td>$\theta = 22.5^\circ$</td>
<td>1.25966939418122</td>
<td>0.692401765581553</td>
<td>0.311042099634158</td>
<td>0.878309721281657</td>
</tr>
<tr>
<td>$\theta = 45^\circ$</td>
<td>1.25966940023470</td>
<td>0.692401766182308</td>
<td>0.311042099422142</td>
<td>0.87830972142706</td>
</tr>
<tr>
<td>$\theta = 67.5^\circ$</td>
<td>1.25966939799363</td>
<td>0.692401766822308</td>
<td>0.311042096992322</td>
<td>0.878309721046069</td>
</tr>
<tr>
<td>$\theta = 90^\circ$</td>
<td>1.25966939718648</td>
<td>0.692401766822308</td>
<td>0.311042096992322</td>
<td>0.878309721295912</td>
</tr>
</tbody>
</table>

Table 7.4: Volumetric flow rate $Q$ at different pipe sections for relative residual tolerance of $\text{tol}= 1 \times 10^{-9}$.

<table>
<thead>
<tr>
<th>section</th>
<th>$Q(t = T/4)$</th>
<th>$Q(t = T/2)$</th>
<th>$Q(t = 3T/4)$</th>
<th>$Q(t = T)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>inlet</td>
<td>1.25966937926702</td>
<td>0.692401765602920</td>
<td>0.311042098160756</td>
<td>0.878309721297001</td>
</tr>
<tr>
<td>$\theta = 0^\circ$</td>
<td>1.25966937926888</td>
<td>0.692401765603025</td>
<td>0.311042098160560</td>
<td>0.878309721297080</td>
</tr>
<tr>
<td>$\theta = 22.5^\circ$</td>
<td>1.25966937926996</td>
<td>0.692401765603258</td>
<td>0.311042098160684</td>
<td>0.878309721297006</td>
</tr>
<tr>
<td>$\theta = 45^\circ$</td>
<td>1.25966937927141</td>
<td>0.692401765603634</td>
<td>0.311042098160667</td>
<td>0.878309721297694</td>
</tr>
<tr>
<td>$\theta = 67.5^\circ$</td>
<td>1.25966937927157</td>
<td>0.692401765604049</td>
<td>0.311042098160491</td>
<td>0.878309721296465</td>
</tr>
<tr>
<td>$\theta = 90^\circ$</td>
<td>1.25966937927269</td>
<td>0.692401765604829</td>
<td>0.311042098160013</td>
<td>0.878309721295912</td>
</tr>
</tbody>
</table>

results for the tolerance level of $\text{tol}= 1 \times 10^{-5}$ and $1 \times 10^{-9}$ are shown in table 7.3 and 7.4 respectively. For tolerance of $1 \times 10^{-5}$, the discrete mass is conserved for at least 7 digits, and for $1 \times 10^{-5}$ the discrete mass is conserved for 11 digits. This shows that the proposed scheme can enforce the discrete continuity to the order of machine accuracy.

7.5 Conclusion

We presented a symmetric finite difference discretization for the Poisson-Neumann problem. The hydrostatic pressure mode is the only vector in the null space of the coefficient matrix, and thus the solvability condition is automatically satisfied. The Neumann boundary condition is naturally incorporated into the coefficient matrix. We solved an analytical problem and demonstrated that the proposed discretization is second order accurate in $l_2$-norm and first order accurate in $l_\infty$-norm. Superior accuracy of the discretization on highly curvilinear and non-smooth grids is presented. We solved the pulsatile incompressible flow in a $90^\circ$ bend where the proposed discretization of the Poisson-Neumann problem
was integrated into a fractional step solver. The comparison of the numerical solution with experimental measurements shows very good agreement.
Figure 7.10: Streamwise velocity at symmetry plane at time $t = T/2$. 

(a) $\theta = 0^\circ$  
(b) $\theta = 22.5^\circ$  
(c) $\theta = 45^\circ$  
(d) $\theta = 67.5^\circ$  
(e) $\theta = 90^\circ$
Figure 7.11: Streamwise velocity at symmetry plane at time $t = T$. 
References


Vita

Hessam Babaee was born in Tehran, Iran. He obtained the B.Sc. degree in Mechanical Engineering from University of Tehran in 2003, and the M.Sc. in Mechanical Engineering from the same university in 2006. He is currently a candidate for the Doctor of Philosophy degree in Mechanical Engineering at Louisiana State University which will be awarded in August 2013. He is also a candidate for M.Sc. in Applied Mathematics at Louisiana State University which will also be awarded in August 2013.