I, Amit Kasliwal,

hereby submit this work as part of the requirements for the degree of:

Master of Science

in:

Aerospace Engineering

It is entitled:


This work and its defense approved by:

Chair: Dr. Karman Ghia

Dr. Urmila Ghia

Dr. Prem Khosla
Flow Separation Control for Cylinder Flow and Cascade Flow Using Vortex Generator Jets

A thesis submitted to the
Division of Research and Advanced Studies
of the University of Cincinnati

in partial fulfillment of the
requirements for the degree of

MASTER OF SCIENCE

in the Department of Aerospace, Engineering and Engineering Mechanics
Of the College of Engineering

Advisor: Dr. Karman Ghia

January 2006

by

Amit Kasliwal

B. Tech., Indian Institute of Technology, Kanpur, 2002

Defense Committee:
Dr. Karman Ghia
Dr. Urmila Ghia
Dr. Prem Khosla
ABSTRACT

Many attempts have been made by researchers, worldwide, to comprehend the physics of separated flows. Study of flow separation is vital as it is encountered in many engineering applications, and is generally detrimental. One such example is flow through a low pressure turbine (LPT) cascade, at relatively low-Re values, where flow separates on the suction surface of the LPT blade, and adversely affects the efficiency of the aircraft engine. Contemporary research is focused on understanding the physics of the separated flow, and identifying control strategies to delay or, if possible at all, prevent the flow separation phenomenon.

The main objective of the present research is to study a model separated flow, and identify a control strategy, which can subsequently be applied to manage the flow in the LPT cascade. To achieve this, a model problem of flow past a circular cylinder is considered, as the geometry for this flow is simple and facilitates a focus on the flow itself. Despite of its simple geometry, the flow past a circular cylinder exhibits a variety of complex flow features which make this a challenging problem to solve. As a validation study, the flow for Re = 3,900 is simulated, and the results obtained are compared with the numerical and experimental data available in the literature. For the flow control study, a baseline solution for flow past a circular cylinder at Re = 13,400 is obtained as a first step towards implementation of flow separation control for preventing or delaying the flow separation. The Re value of 13,400 ensures laminar separation and serves to approximate the flow conditions prevailing in a LPT cascade. Later, flow control is
incorporated by employing vortex generator jets (VGJs) on the upper surface of the cylinder at about 75° from the stagnation point. The jets are issued into the flow with a blowing ratio of 2.0 and are pitched and skewed by 30° and 70°, respectively. A non-dimensional pulsation frequency $F^+$ of 1.0 is used, along with 50% duty cycle. With this understanding, VGJs are then incorporated for the LPT cascade flow. VGJs are placed in a range of 63.5% to 67% $C_{ax}$. All the jet parameters, i.e., blowing ratio, pitch angle, skew angle and duty cycle ratio, are kept the same as for the cylinder case, while the $F^+$ value of 2.33 is employed for the LPT cascade problem.

The three-dimensional, unsteady, full Navier-Stokes equations are solved to obtain accurate prediction of unsteady separated flows governed by the Navier-Stokes (N-S) equations. A fourth-order accurate, compact-difference scheme is used for spatial discretization, with sixth-order filtering to minimize the oscillations in the flow solution. For the cylinder, a multi-block structured grid generated using the grid generation software, GRIDGEN, is used for the present numerical analysis. The grid contains approximately 3.9M grid points, and approximately 70% of the total grid points are concentrated in the wake region to capture the small scales that are expected to exist in this region. A MPI-based higher-order, Chimera version of the FDL3DI flow solver developed by the Air Force Research Laboratory at Wright Patterson Air Force base is used for the numerical computations. PEGSUS a NASA Ames research code is used for storing the connectivity data at the block interfaces.

The baseline case for the cylinder flow at $Re = 13,400$ displays a wide range of vortical structures in the wake region. The separating shear layers are subject to spanwise
instability which leads to the formation of an unsteady and three-dimensional wake, with the characteristic features of typical turbulent flow. It is observed that after the jets are being turned on, the pressure on the surface of the cylinder redistributes in a way so as to reduce the pressure drag significantly. The total pressure loss coefficient and momentum thickness are calculated in the wake at $x/D = 3.0$ and $x/D = 5.0$, and are found to reduce by 10% and 30%, respectively. The flow control simulation for the LPT cascade flow reveals 27% reduction in total pressure loss coefficient, along with the total elimination of separation upon application of VGJs.
Acknowledgements

First and foremost, I am thankful to my advisors Dr. Karman Ghia and Dr. Urmila Ghia, for their continuous guidance and inspiration throughout the course of this research work. Their daily support, discussions have been critical to the success of this research. I would like to thank Dr. Kumar Vemaganti for granting me access to use his cluster to run jobs and for many helpful discussions to understand the parallelization aspects of the flow solver. I would also like to thank Dr. Prem Khosla for reviewing my thesis and agreeing to be on the defense committee.

I would like to acknowledge my colleagues Devkinandan Tokekar, Valentina Kaloyanova, Arun Prakash Raghupathy, Shirdish Poondru, Navtej Singh, Marshall Galbraith, Pavan Kumar Mutnuri, Vijay Narayanan, Rangarajan Sivaji and Soujanya Marisarla for their help at various times.

Finally, I am forever indebted to my parents and sisters for their constant motivation, affection and support while I have been away from home. I could not have done this without them.
# TABLE OF CONTENTS

**LIST OF FIGURES** ........................................................................................................................1

**LIST OF TABLES** ......................................................................................................................... 4

**NOMENCLATURE** .......................................................................................................................... 4

## CHAPTER 1. INTRODUCTION .................................................................................................10

1.1 **PREVIOUS STUDIES ON SEPARATED FLOWS AND ON FLOW SEPARATION CONTROL** ........ 10

1.2 **MULTI-BLOCK STRUCTURED GRIDS** .................................................................................. 11

1.3 **MOTIVATION** .......................................................................................................................... 13

1.4 **OBJECTIVES OF PRESENT RESEARCH** ................................................................................ 13

1.5 **ORGANIZATION OF THE THESIS** ...................................................................................... 15

## CHAPTER 2. MATHEMATICAL FORMULATION .............................................................17

2.1 **PROBLEM SPECIFICATIONS** .............................................................................................. 17

2.2 **GOVERNING EQUATIONS** .................................................................................................... 18

2.3 **COORDINATE TRANSFORMATION** .................................................................................... 21

2.4 **IMPLICIT LARGE-EDDY SIMULATION (ILES)** ................................................................. 25

## CHAPTER 3. GRID GENERATION ......................................................................................27

3.1 **GRID GENERATION** ............................................................................................................. 27

3.2 **INTER-BLOCK COMMUNICATION** ...................................................................................... 28

3.3 **CREATING OVERLAP REGION AT THE BLOCK INTERFACE** ............................................. 30

## CHAPTER 4. NUMERICAL METHOD ..................................................................................33

4.1 **NUMERICAL SCHEME – BEAM AND WARMING SCHEME** ............................................ 33

4.1.1 **LINEARIZATION OF NON-LINEAR TERMS** .................................................................. 34

4.1.2 **APPROXIMATE FACTORIZATION SCHEME OF BEAM AND WARMING (1976) AND
DIAGONALIZATION OF PULLIAM AND CHAUSSEE (1981)** .................................................. 36

4.2 **HIGHER-ORDER ACCURATE COMPACT DIFFERENCING FOR SPATIAL DISCRETIZATION** .... 39

4.3 **FILTERING** ............................................................................................................................ 46
CHAPTER 5. RESULTS AND DISCUSSION ................................................................. 50

5.1. CIRCULAR CYLINDER AT Re = 3,900 ................................................................. 51
5.2. CIRCULAR CYLINDER AT Re = 13,400 ............................................................ 54
5.3. CIRCULAR CYLINDER AT Re = 13,400 WITH GRID REFINEMENT .................... 56
5.3.1. VORTICITY CONTURS ................................................................................. 56
5.3.2. TIME HISTORY OF LIFT AND DRAG COEFFICIENT .............................. 57
5.3.3. Q-CRITERION .............................................................................................. 57
5.3.4. WAKE PROFILE COMPARISON FOR COARSE AND FINE GRIDS ............... 58
5.4. FLOW SEPARATION CONTROL FOR CIRCULAR CYLINDER AT Re = 13,400 ...... 59
5.4.1. VGJ HOLE GEOMETRY AND PARAMETERS ............................................ 59
5.4.2. LIFT AND DRAG COEFFICIENT TIME-HISTORIES ................................. 61
5.4.3. CP VARIATION ALONG THE SURFACE OF THE CYLINDER ..................... 62
5.4.4. ANIMATION OF VORTICITY CONTURS .................................................... 63
5.4.5. MOMENTUM THICKNESS AND TOTAL PRESSURE ................................. 63
5.4.6. EFFICIENCY OF CONTROL ................................................................. 64
5.5. SEPARATION CONTROL FOR LPT AT Re = 25,000 ...................................... 65
5.5.1. CP VARIATION ALONG THE SURFACE OF THE LPT BLADE ................... 67
5.5.2. TOTAL VELOCITY COMPARISON ............................................................ 68
5.5.3. VORTICITY CONTURS AND CP DISTRIBUTION ON BLADE’S SURFACE ....... 68
5.5.4. STREMWISE VELOCITY FLUCTUATIONS ................................................ 69
5.5.5. TOTAL PRESSURE ..................................................................................... 69

CHAPTER 6. SUMMARY, CONCLUSIONS AND RECOMMENDATIONS .............. 70

6.1. SUMMARY AND CONCLUSIONS .................................................................... 70
6.2. RECOMMENDATIONS FOR FUTURE WORK .................................................. 72

References .............................................................................................................. 73
Figures .................................................................................................................... 77
LIST OF FIGURES

Figure 1. Snapshot of the 8-block structured grid used for Re₀ = 3,900........ 77
Figure 2. Coefficient of pressure on the upper surface of the cylinder (Spanwise and time-averaged over 50 characteristic time units)...... 78
Figure 3. Streamwise velocity on the centerline in the wake region (spanwise- and time-averaged)........................................... 79
Figure 4. Streamwise velocity profiles at three locations in the wake region (spanwise and time-averaged)........................................... 80
Figure 5. Crossflow velocity profiles at three locations in the wake region (spanwise and time-averaged)........................................... 81
Figure 6. Instantaneous z-vorticity at mid z-plane, after 200 characteristic time units for Re =3,900...................................................... 82
Figure 7(a). Contours of spanwise Reynolds stress for Re = 3,900............. 82
Figure 7(b). Contours of Turbulence kinetic energy for Re = 3,900........... 83
Figure 8(a). Probe locations where time history is tracked for Re = 3,900..... 83
Figure 8(b). Streamwise velocity time history for Re =3,900................ 84
Figure 9. Contours of vorticity magnitude for Re = 13,400, depicting formation of Karman Vortex street................................. 84
Figure 10. Contours of spanwise Reynolds stress for Re = 13,400.............. 85
Figure 11. Contours of Turbulence kinetic energy for Re = 13,400........... 86
Figure 12(a). Streamwise velocity profiles at three locations in the wake region (spanwise and time-averaged)................................. 87
Figure 12(b). Crossflow velocity profiles at three locations in the wake region (spanwise and time-averaged)................................. 88
Figure 13(a). Streamwise velocity contours for Re = 13,400 after 200 characteristic times................................................................. 89
Figure 13(b). Spanwise velocity contours for Re = 13,400 after 200 characteristic times................................................................. 89
Figure 13(c). Contours of positive (upper) and negative (lower) crossflow velocity Re = 3,900 after 200 characteristic time units.............. 90
Figure 14. Coefficient of pressure on the upper surface of the cylinder (spanwise and time-averaged over 50 characteristic time units)..... 91
Figure 15. Streamwise velocity on the centerline in the wake region (spanwise and time-averaged)......................................................... 92
Figure 16. Snapshot of the fine-grid showing the block-layout and the grid used for the cylinder simulation.................................................. 93
Figure 17. Instantaneous z-vorticity contours for Re = 13,400, after 20 characteristic time................................................................. 94
Figure 18. Grid overlaid on z-vorticity contours for the coarse grid and fine grid (Re = 13,400)............................................................... 95
Figure 19. Instantaneous vorticity magnitude contours for Re = 13,400 from 0.5 to 10................................................................. 96
Figure 20. Lift coefficient time history for cylinder flow at Re = 13,400....... 97
Figure 21. Q-criterion isosurfaces for cylinder flow at Re = 13,400 (mention levels)............................................................... 98
Figure 22. Affect of grid-refinement on streamwise velocity profiles at three
locations in the wake region (spanwise and time-averaged), for Re = 13,400

Figure 23. Affect of grid-refinement on crossflow velocity profiles at three locations in the wake region (spanwise and time-averaged), for Re = 13,400

Figure 24. Jet orientation and parameters

Figure 25. Comparison of coefficient of drag for controlled and uncontrolled case

Figure 26. Comparison of coefficient of lift for controlled and uncontrolled case

Figure 27. Time history of pressure drag and viscous drag coefficient for controlled and uncontrolled flow

Figure 28. Comparison of time- and spanwise-averaged surface coefficient of pressure for controlled and uncontrolled case

Figure 29. Total pressure profile at x/D = 3, for controlled and uncontrolled case for circular cylinder

Figure 30. Total pressure profile at x/D = 5, for controlled and uncontrolled case for circular cylinder

Figure 31. $U_{avg} (1 - U_{avg})$ variation with $y$, at x/D = 5, for controlled and uncontrolled case for circular cylinder

Figure 32. Flow configuration for flow through LPT linear cascade

Figure 33. $U_{avg} (1 - U_{avg})$ variation with $y$, at x/D = 5, for controlled and uncontrolled case for circular cylinder

Figure 34(a). Multi-block structured block layout of the computational domain with 4 point overlaps

Figure 34(b). Multi-block structured grid in the computational domain

Figure 35. Comparison of time- and spanwise-averaged surface coefficient of pressure for present simulation, with experimental results of Huang

Figure 36. Comparison of time- and spanwise-averaged surface coefficient of pressure for present simulation, with experimental results of Huang after shifting

Figure 37. Comparison of velocity magnitude profile for present simulation at x/D = 0.75, with numerical results of Rizzetta

Figure 38. Comparison of velocity magnitude profile for present simulation at x/D = 0.85, with numerical results of Rizzetta

Figure 39. Time- and spanwise-averaged z-vorticity contours for Re = 25,000 (uncontrolled case)

Figure 40. Time- and spanwise-averaged z-vorticity contours for Re = 25,000 (controlled case)

Figure 41. Comparison of surface coefficient of pressure for controlled and uncontrolled case

Figure 42. Contours of streamwise velocity fluctuation (uncontrolled case)

Figure 43. Contours of streamwise velocity fluctuation (controlled case)
LIST OF TABLES

Table I. Coefficients for the interior points…………………………………40
Table II. Boundary coefficients for point 1…………………………………42
Table III. Boundary coefficients for point 2…………………………………43
Table IV. Coefficients for filter formula at interior points………………….46
## NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a_1$, $a_2$, $a_3$</td>
<td>direction cosines</td>
</tr>
<tr>
<td>$C_{ax}$</td>
<td>axial chord length</td>
</tr>
<tr>
<td>$C_p$</td>
<td>pressure coefficient, $2(P_t - P_{ref})/P_{ref}U_{ref}^2$</td>
</tr>
<tr>
<td>$e$</td>
<td>specific internal energy</td>
</tr>
<tr>
<td>$E_t$</td>
<td>total specific energy</td>
</tr>
<tr>
<td>$F, G, H$</td>
<td>Inviscid flux vectors</td>
</tr>
<tr>
<td>$F_v, G_v, H_v$</td>
<td>Viscous flux vectors</td>
</tr>
<tr>
<td>$F_I, G_I, H_I$</td>
<td>Viscous and Inviscid flux vectors</td>
</tr>
<tr>
<td>$i, j, k$</td>
<td>stream-wise, normal and span-wise counters/mesh indices</td>
</tr>
<tr>
<td>$L$</td>
<td>reference length</td>
</tr>
<tr>
<td>$L_r$</td>
<td>Length of recirculation region behind the cylinder</td>
</tr>
<tr>
<td>$M$</td>
<td>Local Mach number</td>
</tr>
<tr>
<td>$M_{ref}$</td>
<td>Reference Mach number</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$p_t, P_t$</td>
<td>non-dimensional total pressure</td>
</tr>
<tr>
<td>$P_{ti}$</td>
<td>non-dimensional total pressure at the inlet</td>
</tr>
<tr>
<td>$P_{to}$</td>
<td>non-dimensional total pressure at the outlet</td>
</tr>
<tr>
<td>$p$</td>
<td>non-dimensional static pressure</td>
</tr>
<tr>
<td>$Q$</td>
<td>vector of dependant variables</td>
</tr>
<tr>
<td>$R$</td>
<td>Universal gas constant</td>
</tr>
<tr>
<td>$R(\cdot, \cdot)$</td>
<td>Residual</td>
</tr>
</tbody>
</table>
Reynolds Number, $\frac{\rho_{\text{ref}} U_{\text{ref}} C_{\text{ax}}}{\mu_{\text{ref}}}$

stream-wise, normal and span-wise components of velocity

time

non-dimensional total temperature

non-dimensional static temperature

Characteristic time

non-dimensional Cartesian velocity components in x, y, z directions

non-dimensional Cartesian velocity components in r, θ, z directions

contravariant velocity components

Time- and Spanwise-averaged values of A

non-dimensional Cartesian coordinates

inlet flow angles

Pitch angle

Skew angle

Angular location of jet from stagnation point

generalized coordinates

metric coefficients of the coordinate transformation

shear stress tensor

non-dimensional fluid density
\( \mu \) non-dimensional dynamic viscosity coefficient

\( \delta_{ij} \) Kronecker delta function (\( \delta_{ij} = 1 \) if \( i = j \) and \( \delta_{ij} = 0 \) if \( i \neq j \))

**II. Subscript Description**

- \( \infty \) free-stream conditions
- \( x, y, z, t \) partial derivatives with respect to the Cartesian coordinates
- \( \xi, \eta, \zeta, \tau \) partial derivatives with respect to the transformed coordinates
- \( r, \theta, z, t \) partial derivatives with respect to the cylindrical coordinates
- \( ax \) axial direction of the geometry
- \( \text{ref} \) reference quantity
- \( V \) viscous terms
- \( I \) inviscid terms

**III. Superscript Description**

- \( * \) dimensional quantity
- \( n, n+1 \) time-levels
- \( T \) transpose of a matrix

**IV. Operators Description**

- \( O () \) Order of

**V. Abbreviations Description**

- \( \text{BR} \) Blowing Ratio
- \( C_w \) Total Pressure Loss coefficient
- \( \text{CPU} \) Central Processing Unit
- \( F^+ \) Non-dimensional pulsation frequency
- \( \text{LES} \) Large Eddy Simulation
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>LHS</td>
<td>Left Hand Side</td>
</tr>
<tr>
<td>LPT</td>
<td>Low-Pressure Turbine</td>
</tr>
<tr>
<td>N-S</td>
<td>Navier-Stokes</td>
</tr>
<tr>
<td>RHS</td>
<td>Right Hand Side</td>
</tr>
<tr>
<td>T.E.</td>
<td>Truncation Error</td>
</tr>
<tr>
<td>VGJs</td>
<td>Vortex Generator Jets</td>
</tr>
<tr>
<td>1D</td>
<td>one-dimensional</td>
</tr>
<tr>
<td>2D</td>
<td>two-dimensional</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

Flow separation is encountered in many engineering applications and it is generally detrimental. During the past few decades, extensive experimental and computational research has been carried out on separated flows to gain a better physical understanding about the flow separation process. One of the engineering applications where flow separation is encountered and is undesirable is the flow in a low-pressure turbine cascade. The lower Reynolds numbers prevailing in the LPT present an acute problem for gas turbine engines which are typically used in many aircrafts. In low-pressure turbines, a significant drop in efficiency is observed between take-off conditions and cruise conditions, due to the associated drop in Reynolds number. At this lower value of Re, the boundary layers on the LPT blades are predominantly laminar, and this makes them susceptible to flow separation on the suction-side of the LPT blade, near the aft portion of the blade. This leads to higher losses, lower stage efficiency and higher fuel consumption. Thus, if separation can be avoided, performance and efficiency of the aircraft as a whole can be improved significantly. Altering the blade geometry to avoid low-Re separation is not a desirable solution as it may affect the engine’s performance at high Re values. An ideal strategy is one which can be turned on at low-Re values and can be turned off at high-Re values i.e., active flow separation control strategy. Therefore, development of a suitable methodology for flow separation control is very important for improving the operating efficiency and performance of the aircraft. Contemporary research is focused on developing active flow separation control strategies to delay, or if possible, to eliminate the flow separation.
1.1 Previous Research work on separated flows and its control

Separated flows have been studied extensively, both numerically and experimentally and a large amount data is available in the literature for a variety of cases on flow separation. One common flow configuration for study of flow separation is flow over a circular cylinder. The fact that despite of the simple geometry of cylinder, flow past a cylinder exhibits complex flow features makes it a very attractive test case. A number of results are available for flow past a cylinder at Reynolds number of 3,900, e.g., the numerical study by Beaudan and Moin (1994), Mittal and Moin (1997) and Kravchenko and Moin (1998), and Morgan, Visbal and Rizzetta (2002) and the experimental work of Ong and Wallace (1996).

Many experimental studies employing flow control strategies have also been carried out in an effort to delay the separation process. Lake et al. (1999) employed passive techniques to delay the boundary-layer separation by modifying the turbine blade geometry. The blades were tested with normal as well as modified surfaces for a range of Reynolds numbers and turbulence intensities. Dimples were recessed into the blade suction surface in order to maintain attached flow, and these researchers found that the effect of dimples is greatest at the lowest Reynolds number where the loss coefficient is reduced by 58%, but due to increase in viscous losses, dimples may not be as advantageous at higher Reynolds number.

In a recent experimental study, Borgeson (2002) employed Micro- Electro-Mechanical Systems (MEMS) devices to control boundary-layer separation for flow over a circular cylinder. Another effective approach used for active control of separation consists of
employing VGJs. These jets are mounted on the surface, and produce streamwise vortices. VGJs produce counter-rotating vortex pairs that provide for momentum transfer across the boundary layer and prevent separation (Johnston and Nishi (1990)). Experiments by Bons, Sondergaard and Rivir (2002) at AFIT reported drastic reduction of suction surface boundary layer separation at low-Re values, and it was observed that the VGJs produced no significant adverse effects when employed at higher (non-separating) Reynolds numbers. It was concluded that the application of VGJs for low Reynolds number separation control on LPT blades shows great promise. Rizzetta et al. recently conducted a numerical study for flow separation control using VGJs on an LPT blade, and a 22% decrease in the total pressure wake loss coefficient was observed. Also, the results revealed a greater distance over which the boundary layer remained attached and a reduced vertical extent of the separated region for the flow control case.

Huang et al. (2003) studied the control of separation that occurs over turbine blades using phased plasma actuators. The effects of Reynolds number and free-stream turbulence levels were analyzed on the onset of separation and reattachment. They observed that the location of separation is relatively insensitive to the experimental conditions; however, the reattachment was found to be very sensitive to the turbulent level and Reynolds number. They concluded that the performance obtained from plasma actuators was comparable to that obtained with the vortex generators.

For efficient operation of an aircraft engine reduction of separation in LPT cascade is extremely important and the present study will contribute significantly towards evaluating the VGJ strategy for improving turbine efficiency.
## 1.2 Structured Grids

The governing non-linear Navier Stokes equations generally cannot be solved analytically, and must employ approximate methods for this solution. Finite-difference, finite-volume and finite-element discretization methods are the most widely and successfully used approximation techniques. All these methods require the computational domain or flow field of interest to be discretized. This process of domain discretization plays a vital role in obtaining an accurate numerical solution for realistic fluid flows. The process of domain discretization by a set of points or nodes over the flow field or domain of interest is called grid generation. As the availability of computational resources advance from day to day, researchers raise their expectations in simulating the realistic fluid flows more and more accurately. Over the past few decades, CFD researchers have been focusing on grid generation as a core branch of CFD, as a good grid plays a vital role in improving the fidelity of the solution. The governing partial differential equations are to be solved at every grid point to obtain an accurate solution representing the flow physics of the fluid simulated numerically. The most widely used grids are structured and unstructured. Unstructured grids are relatively easier to generate and takes less computational time as compared to structured grid. But developing the solution algorithm for an unstructured grid requires a much more complex solution data structure as compared to the structured grid. It is easier to obtain a higher-order accurate solution on a structured grid as compared to an unstructured grid. Also structured grids will find their added advantage in the need of representing surface-normal-derivative boundary condition and in some turbulence models where wall normal distance plays a vital role in computations.
In the structured grid generation technique, the set of points discretizing the flow domain will form the vertices of the quadrilateral cell in the case of a 2-D computational domain or a hexahedral cell for a 3-D case. In a structured grid, the domain is discretized in an orderly manner with some regular pattern. Whereas, in an unstructured grid, this regular pattern is absent, and the domain is generally discretized as triangles or tetrahedra in 2-D and 3-D.

Equations governing the flow must be satisfied with the possible highest practical accuracy, in order that the numerical simulation represents the physics of the fluid flow, whereas the problem of grid generation is not governed by any physical laws or conservation principles. Therefore, the governing grid generation equations need not be satisfied exactly. This allows researchers an additional degree of freedom to experiment with the techniques of grid generation.

The governing non-linear equations are difficult to solve directly without any numerical approximation. The grid used to solve these approximated equations has a vital role to play in reducing the errors associated with these assumptions. To fulfill this role, the generated grid must be sufficiently dense within the practical limits of computational resources. The grid spacing should be varying smoothly and sufficiently refined to capture the changes in the gradients of the solution. The accuracy of a numerical simulation can be impaired, if there is a discontinuity in the grid or the grid is highly skewed. If the generated mesh is a boundary-conforming curvilinear grid, it will aid in representing the boundary conditions more accurately and easily. In some cases, the governing partial differential equations can be simplified based on grid-related approximation and can be solved with relatively great ease.
The structured grid can be generated algebraically by interpolation from boundary points, e.g., transfinite interpolation, or by solving a set of partial differential equations in the region, e.g., elliptic grid generation. Each technique has its own pros and cons; algebraic grid generation is relatively faster than generating grids by solving partial differential equations, whereas the grids generated by solving partial differential equations are generally smoother compared to algebraically generated grids.

1.3 Motivation

As discussed earlier, prevention of flow separation is important for reducing the losses occurring in a low-pressure turbine stage and for developing a suitable flow control technique. The present study uses high-order compact difference schemes as implemented in the flow solver FDL3DI. Also, the present study focuses on usage of multi-block structured grid approach to benefit from the parallelization flow solver using the available massively parallel computing systems. Hence, reducing the total physical turn around time required to complete a simulation significantly. All the simulations conducted in the present study are 3-D to match closely to the realistic application. FDL3DI is a research code developed by Air Force Research Laboratory at Wright Patterson Air Force Base is used as the flow solver. Among the several versions that are available the high-order accurate, parallel, Chimera, LES version of FDL3DI is used in the present work.

1.4 Objectives of Present Research

The main objective of the present research is to develop a suitable flow separation control strategy for flow through the LPT linear cascade, using a higher-order accurate analysis as implemented in high-order accurate, Chimera, LES version of FDL3DI.
Multi-block structured grid approach is selected in the present study as it helps in significant reduction of the physical turn around time for completing a simulation. Flow separation control is studied for cylinder flow and LPT cascade configurations. For the cylinder flow simulations, a multi-block structured grid with higher concentration of points in the wake region is used. For the LPT cascade flow, multiple topologies are used owing to the complexity of the geometry and the flow. In order to achieve a higher-order accurate solution using multi-block structured grid approach for this problem, various cases are solved, with the specific objectives mentioned below:

(i) To understand working with the flow solver and usage of multi block structured grid:
   - Flow past a circular cylinder at Re = 3,900

(ii) To provide the baseline solution for cylinder flow:
    - Simulation for flow past a circular cylinder at Re = 13,400

(iii) To test the implementation of flow separation control using VGJs:
    - Flow separation control simulation using VGJs is performed for flow past a circular cylinder at Re = 13,400

(iv) To provide a baseline solution for the cascade flow:
    - Numerical results are obtained for the cascade flow at Re = 25,000

(v) To control the flow separation for cascade flow using VGJs
    - Flow separation control simulation using VGJs is performed for flow the cascade flow at Re = 25,000

All the simulations are three-dimensional and are simulated using the Implicit Large-Eddy Simulation Technique (Morgan and Visbal (2002)).
1.5 Organization of the Thesis

In this Thesis, Chapter 1 gives an introduction to the topic of the present research and Chapter 2 provides the mathematical formulation of the problem. Chapter 3 provides the details of grid generation. The details of numerical scheme used in the flow solver are included in Chapter 4. Chapter 5 provides the results and discussion, and is followed by summary, conclusions and recommendations for future work in Chapter 6. Finally, references and figures are included.
Chapter 2

Mathematical Formulation

This chapter presents the mathematical formulation of the problem. The formulation used here was developed at the Air Force Research Laboratory (AFRL) at Wright Patterson Air Force Base (WPAFB). For the sake of completeness, a brief outline of the methodology is presented in this chapter. The governing equations used in the present study are the compressible, unsteady, three-dimensional Navier-Stokes (N-S) equations. These equations are written in non-dimensional variables and cast in the strong conservation-law form. This chapter consists of four subsections, presenting a detailed description of the mathematical formulation used in the present study. The first section gives a brief description about the problem specification/flow configuration. The second section discusses the governing N-S equations in the Cartesian coordinate system and the non-dimensionalization procedure applied to these equations. The third section explains the transformation from a Cartesian coordinate system \((x,y,z)\) to a general time-dependent curvilinear coordinate system \((\xi,\eta,\zeta)\). The final sub-section describes the implicit Large Eddy Simulation technique.

2.1. Problem Specifications

During the present study two different geometries, circular cylinder and low-pressure turbine blade are considered for the analysis. For the case of the cylinder, a cylinder non-dimensional diameter \(D\) of 1.0 unit is considered at \(Re = 13,400\). For the study of flow through the LPT blade linear cascade, simulations were carried out for a cascade using a Pratt & Whitney PAK-B low-pressure turbine blade, which is extruded from a 2-D profile to represent a 3-D geometry. Figure 33 shows the flow
configuration with PAK-B low-pressure turbine blade profile. At the inflow boundary, the flow enters at an angle of 35° to the axial direction, and the exit angle is -60°, hence, the total flow turning angle is 95°. The Pak-B blade is highly curved. Initially, for validation study a test case of flow over circular cylinder has been considered.

2.2. Governing Equations

The governing equations the compressible, unsteady, three-dimensional N-S equations in non-dimensionalized, vector form and for a Cartesian coordinate system (x, y, z) they are given as

\[
\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0 .
\]

2.1)

Where \( Q \) is the vector of dependent or unknown variables, defined as

\[
Q = [\rho, \rho u, \rho v, \rho w, E,]^	op ,
\]

2.2)

and F, G, and H are the flux vectors, that can be expressed as

\[
F = \begin{bmatrix}
\rho u \\
\rho u^2 + p - \tau_{xx} \\
\rho uv - \tau_{xy} \\
\rho vw - \tau_{xz} \\
\left( E_i + p \right) u - u\tau_{xx} - v\tau_{xy} - w\tau_{xz} + q_x^	op
\end{bmatrix},
\]

2.3)

\[
G = \begin{bmatrix}
\rho v \\
\rho uv - \tau_{yx} \\
\rho v^2 + p - \tau_{yy} \\
\rho vw - \tau_{yz} \\
\left( E_i + p \right) v - u\tau_{yx} - v\tau_{yy} - w\tau_{yz} + q_y^	op
\end{bmatrix},
\]

2.4)

and
\[ H = \begin{bmatrix} \rho w \\ \rho u w - \tau_{xz} \\ \rho v w - \tau_{yz} \\ \rho w^2 + p - \tau_{zz} \\ (E_t + p) w - u \tau_{xz} - v \tau_{yz} - w \tau_{zz} + q_z \end{bmatrix}. \] (2.5)

where \( \rho \) is the density, \( u, v \) and \( w \) are the Cartesian components of velocity, and \( E_t \) is the specific total energy. The total energy of the fluid, \( E_t \) is expressed in terms of internal energy and kinetic energy as

\[ E_t = \rho \left( e + \frac{u^2 + v^2 + w^2}{2} \right). \] (2.6)

The components of shear stress that appear in the expressions for the flux vectors in Eqs. 2.3) - 2.5) are given by

\[ \tau_{xx} = \frac{2}{3} \frac{\mu}{\text{Re}} \left( 2 \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{\partial w}{\partial z} \right), \] (2.7)

\[ \tau_{yy} = \frac{2}{3} \frac{\mu}{\text{Re}} \left( 2 \frac{\partial v}{\partial y} - \frac{\partial u}{\partial x} - \frac{\partial w}{\partial z} \right), \] (2.8)

\[ \tau_{zz} = \frac{2}{3} \frac{\mu}{\text{Re}} \left( 2 \frac{\partial w}{\partial z} - \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} \right), \] (2.9)

\[ \tau_{xy} = \frac{\mu}{\text{Re}} \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right), \] (2.10)

\[ \tau_{xz} = \frac{\mu}{\text{Re}} \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right), \] (2.11)

\[ \tau_{yz} = \frac{\mu}{\text{Re}} \left( \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right). \] (2.12)

The heat flux terms present in Eqs. 2.3) - 2.5) can be written as
\[
q_x = -\frac{\mu}{(\gamma - 1)M_{ref}^2 \operatorname{Re} \operatorname{Pr}} \frac{\partial T}{\partial x}, \quad 2.13
\]

\[
q_y = -\frac{\mu}{(\gamma - 1)M_{ref}^2 \operatorname{Re} \operatorname{Pr}} \frac{\partial T}{\partial y}, \quad 2.14
\]

\[
q_z = -\frac{\mu}{(\gamma - 1)M_{ref}^2 \operatorname{Re} \operatorname{Pr}} \frac{\partial T}{\partial z}, \quad 2.15
\]

Assuming perfect gas, the equation of state is given by

\[
p = (\gamma - 1)\rho e \quad . \quad 2.16
\]

Using the definition of Mach number in Eq. 2.16), temperature can be computed as

\[
T = \frac{\gamma M^2 p}{\rho} \quad . \quad 2.17
\]

The molecular viscosity is computed using Sutherland’s law, given by

\[
\mu = \frac{(1 + \bar{T})}{\bar{T} + T} T^{3/2} \quad . \quad 2.18
\]

where \(\bar{T}\) is Sutherland’s constant, and is equal to \(\frac{110^\circ K}{T_{ref}}\).

The Reynolds number and the Mach number are defined as:

\[
Re = \frac{U_{ref} L_{ref} \rho_{ref}}{\mu_{ref}}, \quad 2.19
\]

and

\[
M = \frac{U_{ref}}{\sqrt{\gamma R T_{ref}}} \quad . \quad 2.20
\]

The following non-dimensional variables are used in the governing equation written in non-dimensional form in the Eq. 2.1),
\[ x = \frac{x^*}{L_{\text{ref}}}, \quad y = \frac{y^*}{L_{\text{ref}}}, \quad z = \frac{z^*}{L_{\text{ref}}}, \quad t = \frac{t^*U_{\text{ref}}}{L_{\text{ref}}}, \]

\[ u = \frac{u^*}{U_{\text{ref}}}, \quad v = \frac{v^*}{U_{\text{ref}}}, \quad w = \frac{w^*}{U_{\text{ref}}}, \]

\[ \rho = \frac{\rho^*}{\rho_{\text{ref}}}, \quad \rho = \frac{p^*}{\rho_{\text{ref}} U_{\text{ref}}^2}, \quad T = \frac{T^*}{T_{\text{ref}}}. \]

where \( \rho_{\text{ref}}, \mu_{\text{ref}}, L_{\text{ref}} \) and \( T_{\text{ref}} \) are the reference quantities for the respective variables.

Except for reference length, for all other variables, the reference conditions used for non-dimensionalisation are taken as the flow conditions at the inlet boundary. For a turbine cascade, the reference length is the axial chord, \( C_{\text{ax}} \), of the blade, and for a circular cylinder, the reference length is the diameter of the cylinder.

### 2.3. Coordinate Transformation

The transformation of the Navier-Stokes equations from a Cartesian coordinate system to a general curvilinear co-ordinate system is discussed in this section. The complex curvilinear geometries like LPT blade can be conveniently described in terms of boundary-aligned coordinates, so that the boundary conditions can be represented accurately, as it is difficult to represent the boundary conditions accurately for such arbitrary shapes, using a Cartesian or a cylindrical co-ordinate system.

The governing equations are transformed from the physical domain \((x,y,z)\) to the computational domain \((\xi, \eta, \zeta)\) using the general transformation of the form

\[ \xi = \xi(x,y,z), \]

\[ \eta = \eta(x,y,z), \]

\[ \zeta = \zeta(x,y,z). \]
The derivation of these transformed equations is explained in detail by Ayyalasomayajula.

After applying the transformation to Eq. 2.1) and rearranging the terms, we have

\[
\frac{\partial Q}{\partial \tau} + \left( \frac{\partial F}{\partial \xi} \frac{\partial \xi}{\partial x} + \frac{\partial F}{\partial \eta} \frac{\partial \eta}{\partial x} + \frac{\partial F}{\partial \zeta} \frac{\partial \zeta}{\partial x} \right)
+ \left( \frac{\partial G}{\partial \xi} \frac{\partial \xi}{\partial y} + \frac{\partial G}{\partial \eta} \frac{\partial \eta}{\partial y} + \frac{\partial G}{\partial \zeta} \frac{\partial \zeta}{\partial y} \right)
+ \left( \frac{\partial H}{\partial \xi} \frac{\partial \xi}{\partial z} + \frac{\partial H}{\partial \eta} \frac{\partial \eta}{\partial z} + \frac{\partial H}{\partial \zeta} \frac{\partial \zeta}{\partial z} \right) = 0.
\]

Equation 2.25) is a complete transformed equation obtained after applying the generalized transformation to the vector form of the N-S equations. The transformed governing N-S equations are expressed in strong conservation form as proposed by Viviand (1974) and Vinokur (1974) by dividing them with the Jacobian of the transformation, and then rearranging by adding and subtracting the like terms.

\[
\left( \frac{Q}{J} \right)_\tau + \left( \frac{F \xi_x + G \xi_y + H \xi_z}{J} \right)_x + \left( \frac{F \eta_x + G \eta_y + H \eta_z}{J} \right)_y + \left( \frac{F \zeta_x + G \zeta_y + H \zeta_z}{J} \right)_z = 0
\]

Defining

\[
Q_1 = \frac{Q}{J}
\]

\[
F_1 = \frac{F \xi_x + G \xi_y + H \xi_z}{J}
\]

\[
G_1 = \frac{F \eta_x + G \eta_y + H \eta_z}{J}
\]

and \( H_1 = \frac{F \zeta_x + G \zeta_y + H \zeta_z}{J} \)

Eq. (2.41) can be written in the strong conservation law form as
\[
\frac{\partial Q_1}{\partial \tau} + \frac{\partial F_1}{\partial \xi} + \frac{\partial G_1}{\partial \eta} + \frac{\partial H_1}{\partial \zeta} = 0 \quad .
\]  

(2.31)

With \( \xi, \eta, \zeta \) and \( \tau \) representing general curvilinear transformed variables, the governing equations in the vector form are shown in Eq. (2.41) where \( Q \) is as defined in the previous section by Eq. (2.2). The terms \( F_1, G_1, H_1 \) consist of the viscous and inviscid terms, which can be written as

\[
\frac{\partial Q}{\partial \tau} + \frac{\partial}{\partial \xi}\left(F_1 - \frac{1}{Re} F_v\right) + \frac{\partial}{\partial \eta}\left(G_1 - \frac{1}{Re} G_v\right) + \frac{\partial}{\partial \zeta}\left(H_1 - \frac{1}{Re} H_v\right) = 0
\]

(2.32)

where \( Q \) is the vector of dependent variables as shown in Eq. 2.2). \( F_1, G_1, H_1 \) are the inviscid flux vectors given as

\[
F_1 = \begin{bmatrix}
\rho U \\
\rho u U + \xi_p \rho \\
\rho v U + \zeta_p \\
\rho w U + \zeta_p \\
\rho E_i U + \zeta_p u_i p
\end{bmatrix},
\]

(2.33)

\[
G_1 = \begin{bmatrix}
\rho V \\
\rho u V + \eta_v \rho \\
\rho v V + \eta_v p \\
\rho w V + \eta_v p \\
\rho E_i V + \eta_v u_i p
\end{bmatrix},
\]

(2.34)

\[
H_1 = \begin{bmatrix}
\rho W \\
\rho u W + \xi_p \rho \\
\rho v W + \zeta_p \\
\rho w W + \zeta_p p \\
\rho E_i W + \zeta_p u_i p
\end{bmatrix},
\]

(2.35)

and \( F_v, G_v \) and \( H_v \) are the viscous flux vectors given by
In Eqs. 2.33) - 2.35), U, V, and W are contra-variant components of velocity. They can be expressed in terms of the Cartesian components of velocity as shown in the following equations

\[ U = \xi_1 + \xi_x u + \xi_y v + \xi_z w , \tag{2.39} \]

\[ V = \eta_1 + \eta_x u + \eta_y v + \eta_z w , \tag{2.40} \]

\[ W = \zeta_1 + \zeta_x u + \zeta_y v + \zeta_z w \tag{2.41} \]

and \( E_t \) is the total energy, which is defined in Eq 2.6). Using the compact notation \( x_i, \ i=1,2,3 \) to represent the x, y and z coordinates, respectively, and similarly, \( \xi_i \) for \( \xi, \eta, \zeta \), the components of shear stress tensor \( \tau_{ij} \) can be written as
\[ \tau_{ij} = \mu \left( \frac{\partial \xi_k}{\partial x_j} \frac{\partial u_i}{\partial \xi_k} + \frac{\partial \xi_k}{\partial x_i} \frac{\partial u_j}{\partial \xi_k} \right) - \frac{2}{3} \mu \delta_{ij} \frac{\partial \xi_l}{\partial x_k} \frac{\partial u_k}{\partial \xi_l} \]  \quad 2.42)

while

\[ b_i = u_j \tau_{ij} - Q_i \]  \quad 2.43)

and

\[ Q_i = -\left[ \frac{1}{(\gamma-1)M_{\text{ref}}^2} \right] \left( \frac{\mu}{Pr} \right) \frac{\partial \xi_j}{\partial x_i} \frac{\partial T}{\partial \xi_j} \]  \quad 2.44)

The Eq. 2.32) represents the unsteady, three dimensional compressible unfiltered Navier-Stokes equations. The equation is expressed in curvilinear coordinates representing a body-fitted coordinate system and is in nondimensional conservative form. The Eq. 2.32) is used as the governing equation for Direct Numerical Simulations (DNS), where all the scales of motion of a turbulent flow ranging from the large energy-containing scales or integral scales to the dissipative scales also called as viscous or Kolmogoroff scales. But this approach is having a limitation on computational resources for the problems with higher Reynolds numbers.

2.4. **Implicit Large-Eddy Simulation (ILES)**

As mentioned earlier, each of the standard SGS models have their own pros and cons. Because of the excessive damping characteristics of these standard SGS models and other difficulties, the researchers group at AFRL considered a different approach called Implicit Large-Eddy Simulation technique, which is somewhat similar to the Monotonically Integrated Large-Eddy Simulation (MILES) approach. They experienced that the results obtained using the ILES technique with no explicit subgrid scale modeling
performed better than the standard LES with constant-coefficient and dynamic Smagorinsky models. Visbal et al. (2002) found that the standard SGS models, unlike the optimized low-pass filter used in FDL3DI, exert dissipation over a wide range of wave numbers including some of the resolved scales. And they predicted that it could be because of the inability of the model to effectively discriminate between the resolved and unresolved scales, and cannot be corrected by simply adjusting the constant in the SGS model.

In the ILES technique, the dissipative properties of the numerical scheme are used in lieu of an explicitly employed standard SGS model. The present flow solver FDL3DI has up to sixth-order accuracy for spatial discretization using compact difference discretization. Like other centered schemes, compact schemes are also non-dissipative in nature and therefore susceptible to numerical instabilities because of the unrestricted growth of high-frequency modes, which will lead to numerical instability. In order to prevent this, a high-order low-pass spatial filtering technique is incorporated in FDL3DI by the researchers at AFRL. This high-order low-pass filtering, up to sixth-order accurate, is used to enhance the dissipative properties of the compact differencing implicitly for the ILES technique.

The following chapter describes the grid generation approach selected for the present study, and some issues associated with this approach.
Chapter 3

Grid Generation

Grid generation and implementation of boundary condition are two main contributors to the accuracy of a numerical solution. The grid plays such a key role that grid generation has, in fact, developed into a branch by itself in the field of CFD. This chapter presents a brief overview on the grid generation approach used in the present study. First section of the chapter provides a discussion about the grid topology selected for the circular cylinder geometry and LPT cascade geometry. The second section describes the inter-block communication of shared faces on massively parallel machines. Third section describes in detail about creating the overlap at the block interfaces.

3.1. Grid Generation

For the problem of flow over circular cylinder it is desired to have a body-fitted or boundary-conforming curvilinear coordinate system so that the boundary conditions can be represented more accurately on the grid. So, an O-grid topology has been selected for this case. Two grids of different densities are used for the study of the cylinder flow configuration and are shown in Fig. 1 and Fig. 16. The first grid is relatively coarser and is generated in Gridgen while a code is developed to generate the second grid. The first grid is relatively straightforward; first, an O-grid is generated and then the blocking is done in tangential direction. The second grid contains a total of four million grid points and bears an O-topology. In order to optimize the total number of grid points, the grid is generated in a way that approximately 70% of the total grid-points are concentrated in the wake region. The blocks containing the cylinder’s surface and the blocks in the wake region have higher concentration of grid points as compared to the other blocks.
For the LPT cascade problem, a periodic boundary condition in the cross flow direction is required to be implemented to incorporate the linear cascade effect in the present numerical study. Due to the highly curved shape of the blade coupled with the necessity to have a periodic boundary condition in the cross flow direction, a single-topology grid results in a highly skewed grid over the blade surface, which will lead to possibility of high truncation error and will affect the fidelity of the solution obtained. Therefore, an O-grid topology in the proximity of the blade and an H-grid topology elsewhere is used for LPT cascade problem as can be seen in Fig. 34. This result in a good quality grid compared to single topology grid. The planes located in the upstream and downstream regions fix the location of the inflow and outflow boundaries.

The grid around the blade is generated using GridPro by Mutnuri (2003), where the domain is decomposed into relatively small near-rectangular blocks; within each block algebraic grid generation technique is used to generate the mesh and using the variational technique, smoothness and continuity of the grid lines at the block interfaces is ensured, resulting in a good quality near-orthogonal multi-block structured grid.

3.2. Inter-Block Communication

The concept of decomposing the computational domain into multiple blocks allows the direct parallelization of the flow solver on parallel systems. The flow solver FDL3DI has been parallelized using Message Passing Interface (MPI) library calls, such that each block is assigned to a different processor. To make sure about the load balancing while running the solver in parallel mode, the computational domain has to be
partitioned in such a way that all the blocks are approximately of same size. Since each block is assigned to a different processor, it necessitates the requirement of explicit communication of flow solutions between the blocks residing on different processors. The exchange of data between neighboring blocks is achieved by using explicit MPI send and receive library calls. A smooth transition of flow solution from one block to another block is achieved by having an overlap region of grid points at the block interface. But having an overlap region means solving the governing equations at additional grid points, thereby resulting in an overhead of computational resources. So, the optimizing feature which dictates the extent of minimum required overlap region is the stencil size used by the numerical scheme in the flow solver. In general, a second-order accurate scheme will require a minimum of two-cell overlap region at the block interface, with one-cell from each of two neighboring blocks. For the present study, a fourth-order accurate compact difference scheme is employed, having three-point stencil for the finite difference representation of the spatial derivatives. This necessitates having a region of at least two-cell overlap at the block interface, with one-cell being shared from each side of the inter-block boundary. In the present study, a four-cell overlap is provided at the block interfaces for both, the cylinder and the cascade flow configurations. Figure 3.1 shows the basic idea of inter-block communication using an overlapped region at the block
interface. The following section will give the details of an overlapping procedure used for the present study.

3.3. Creating Overlap region at the Block Interface

For the case of coarse cylinder grid and the grid for the cascade, the current block is extended by 2 cells into the neighboring block by injecting the grid points. Similarly the neighboring block extends 2 cells into the current block thereby creating the required 4-cell overlap, necessary for the inter-block communication. Unlike the traditional chimera, the grid points are injected in such a way that the extended planes coincide with the grid points on the neighboring block planes and hence actual interpolation is not required. Thus, a four-cell overlap is achieved for the present problem in order to form the five-point stencil for the C6F10 scheme used in the flow solver.

The same concept as described in last section is used to create the overlaps for the fine grid generated for the cylinder case. However, for this grid a few block pairs exist, which have non-equal number of cell at the common interface region. Creating the overlaps between these blocks does not lead to a five-point overlap or a four-cell overlap, in contrast to what was discussed in the last paragraph, because of the difference in the grid densities/cell size of neighboring blocks. The overlaps lead to a six-cell overlap as per

Fig. 3.1. Multi-block diagram showing Block Connectivity (after Joe F. Thompson et al.)
the small cell size and to three-cell overlap as per the large cell size. Since in this overlap not all grid points of neighboring blocks coincide with each other, actual interpolation is required unlike the cases discussed in last paragraph.

Fig. 3.2 shows the process of injecting grid points from one block into the neighboring block and vice versa forming a four-cell overlapped region at the block interface. As shown in this figure mesh 1 is acting as the donor mesh for the planes IL-4 and IL-3 as the information is being transferred from mesh 1 to mesh 2, while mesh2 is acting as the acceptor mesh. But for planes 4 and 5 it is vice versa. Although the grid points from the two blocks in the overlap region are coincident, they have been shown slightly staggered in the Fig. 3.2 for better understanding.

Fig. 3.2. Schematic representation of five-point (four cells) mesh overlap
After creating the overlap, PEGSUS, a NASA Ames Research Center interpolation code, is used to determine the flow variables at the Chimera boundaries between the blocks. This code acts as an interface between the processors in communicating the grid points involved in the overlap region. The connectivity information for each and every block is stored by the PEGSUS in the form of donor and acceptor mesh points in the overlapped region required for the inter-block communication. Data is exchanged between the adjacent sub-domains for every sub-iteration as well as after each application of filter. The flow solution is updated at the Chimera boundaries using the explicit message passing MPI send and receive library calls for the stored connectivity data. Therefore, for every iteration, the present scheme computes the flow solution by solving the governing equations implicitly in the interior region of the sub-domain and by updating the block boundary information explicitly using an overlapped region.
Chapter 4

Numerical Method

The flow solver FDL3DI uses the implicit, approximately factored, Beam and Warming scheme (1976) to solve the governing three-dimensional, unsteady and compressible Navier-Stokes equations. The code has provision for both first- and second-order accurate implicit temporal discretization and for spatial discretization, fourth-order accurate compact differencing is used. The details of the time integration method, including the linearization of the nonlinear terms, the approximate factorization, and the diagonalization are explained in the first section of this chapter. The second section discusses the details of the computational stencil, differencing formulas, and the coefficients involved in the numerical scheme used for the spatial discretization. The last section explains the solution procedure utilized in this flow solver.

4.1. Numerical Scheme

The Beam-Warming scheme belongs to the class of Alternating Direction Implicit (ADI) schemes wherein implicit methods are used for time integration. ADI schemes employ alternating one-dimensional sweeps in each spatial direction in order to efficiently solve the system of algebraic equations obtained after the discretization of the system of governing partial differential equations. Local time linearization is applied to the nonlinear terms, and an approximate factorization of the three-dimensional implicit operator is used to produce locally one-dimensional operators. This results in block-tridiagonal matrices with reduced band width, the inversion of which is comparatively less computationally expensive than the inverse of matrices prior to the approximate factorization. Diagonalization of the equations produces further a computationally
efficient modification that results in scalar tridiagonal or pentadiagonal operators in place of the original block operators. Each of the above mentioned key aspects of the scheme are explained in detail in the coming three sections.

4.1.1. Basic form of the scheme and its Linearization

For simplicity of discussion a two-dimensional nonlinear system of equations is considered, which can be written as

$$\frac{\partial Q}{\partial t} + \frac{\partial E(Q)}{\partial x} + \frac{\partial F(Q)}{\partial y} = 0 . \quad (4.1)$$

In the Beam-Warming scheme, the solution is marched in time using the following difference formula:

$$\Delta^n Q = \frac{\theta_1 \Delta t}{1 + \theta_2} \left( \frac{\partial}{\partial t} (\Delta^n Q) + \frac{\partial}{\partial t} (Q^n) + \frac{\theta_2}{1 + \theta_2} \Delta^{n-1} Q \right)$$

$$+ O\left[ \left( \theta_1 - \frac{1}{2} - \theta_2 \right)(\Delta t)^2 + (\Delta t)^3 \right], \quad (4.2)$$

where $\Delta^n Q = Q^{n+1} - Q^n$. Equation (4.2) is said to be in Delta form. This is a general difference formula from which many of the standard schemes can be reproduced with the appropriate choice of the parameters $\theta_1$ and $\theta_2$. The combination $\theta_1 = 1, \theta_2 = 0$, gives the Euler implicit scheme, which is first-order accurate in time. Using the $\theta_1 = 1, \theta_2 = 1/2$ combination, the three–point backward implicit scheme, which is second-order accurate in time, is obtained. Replacing $\frac{\partial}{\partial t} Q$ by $-\left( \partial_x E(Q) + \partial_y F(Q) \right)$ in Eq. (4.2), we obtain

$$\Delta^n Q = \frac{\theta_1 \Delta t}{1 + \theta_2} \left[ \frac{\partial}{\partial x} \left( -\Delta^n E(Q) \right) + \frac{\partial}{\partial y} \left( -\Delta^n F(Q) \right) \right]$$

$$+ \frac{\partial}{\partial x} \left( -E^n(Q) \right) + \frac{\partial}{\partial y} \left( -F^n(Q) \right) \right]$$

$$+ \frac{\theta_2}{1 + \theta_2} \Delta^{n-1} Q + O\left[ \left( \theta_1 - \frac{1}{2} - \theta_2 \right)(\Delta t)^2 + (\Delta t)^3 \right]. \quad (4.3)$$
Because of the presence of $\Delta^n E(Q)$ and $\Delta^n F(Q)$, this equation is nonlinear $Q^{n+1}$ and needs to be linearized in order to solve for $Q^{n+1}$. Beam and Warming suggested that Taylor series expansions be used for $E^{n+1}$ and $F^{n+1}$ to obtain a linear equation, as follows:

$$E^{n+1}(Q) = E^n(Q) + [A]^n (Q^{n+1} - Q^n) + O[(\Delta t)^2],$$

$$F^{n+1}(Q) = F^n(Q) + [B]^n (Q^{n+1} - Q^n) + O[(\Delta t)^2], \quad (4.4a, b)$$

which can be rewritten as

$$\Delta^n E(Q) = [A]^n \Delta^n Q + O[(\Delta t)^2],$$

$$\Delta^n F(Q) = [B]^n \Delta^n Q + O[(\Delta t)^2]. \quad (4.5a, b)$$

In the above equations, $[A]$ and $[B]$ are defined as

$$[A] = \left[ \frac{\partial E(Q)}{\partial Q} \right], \quad (4.6a)$$

and

$$[B] = \left[ \frac{\partial F(Q)}{\partial Q} \right], \quad (4.6b)$$

and are called the flux Jacobians. Further details about the flux Jacobian matrices are available in Tannehill et al. (1997). Substituting Eq. (4.5) in Eq. (4.3) and rearranging all the terms, we obtain

$$Q^{n+1} \left\{ I + \frac{\theta_1 \Delta t}{1 + \theta_2} \left[ \frac{\partial}{\partial x} [A]^n + \frac{\partial}{\partial y} [B]^n \right] \right\} = Q^n \left\{ I + \frac{\theta_1 \Delta t}{1 + \theta_2} \left[ \frac{\partial}{\partial x} [A]^n + \frac{\partial}{\partial y} [B]^n \right] \right\}$$

$$+ \frac{\Delta t}{1 + \theta_2} \left[ \frac{\partial}{\partial x} (-E^n(Q)) + \frac{\partial}{\partial y} (-F^n(Q)) \right]$$

$$+ \frac{\theta_2}{1 + \theta_2} \Delta^{n-1} Q + O \left( \left( \theta_1 - \frac{1}{2} - \theta_2 \right)(\Delta t)^2 + (\Delta t)^3 \right). \quad (4.7)$$
The left hand side of Eq. (4.7) being at the current time level \( n+1 \) is unknown, and is referred to as the implicit side of the equation. The right hand side of Eq. (4.7) being at the previous time level \( n \) is known, and is referred to as the explicit side of the equation. A direct solution to this equation is avoided owing to the large operation count in treating multi-dimensional systems, which makes it prohibitively time consuming. Therefore, it is desired to reduce the multi-dimensional inversions to a sequence of one-dimensional inversions. This can be done using the method of fractional steps or the method of approximate factorization. The method of approximate factorization proposed by Beam and Warming is discussed briefly in the following sub-section.

4.1.2. Approximate Factorization scheme of Beam and Warming (1976)

It is required to write Eq. (4.7) in delta form before prior to implementing the approximate factorization. This is achieved done by bringing the first term on the RHS of Eq. (4.7) to the LHS and expressing the LHS in delta form as

\[
\left[ I + \frac{\theta_1 \Delta t}{1 + \theta_2} \left( \frac{\partial}{\partial x} [A]^n + \frac{\partial}{\partial y} [B]^n \right) \right] \Delta Q^n = -\frac{\Delta t}{1 + \theta_2} \left( \frac{\partial}{\partial x} E^n (Q) + \frac{\partial}{\partial y} F^n (Q) \right) + \frac{\theta_1}{1 + \theta_2} \Delta Q^{n-1} \left[ \left( \frac{\theta_1}{2} - \theta_2 \right) (\Delta t)^2 + (\Delta t)^3 \right] \]

(4.8)

The order of accuracy of the numerical approximation of the spatial derivatives in the implicit part of Eq. (4.8) significantly affects the required amount of computational work needed to solve this system. For example, if a second-order central difference scheme is used to approximate the spatial derivative, the implicit part of the above equation will lead to a large, banded matrix. According to Pulliam (1993), solving the resulting system of algebraic equations becomes computationally very expensive in spite of the banded
matrix being sparse. In order to reduce the computational effort, Beam and Warming introduced the method of approximate factorization to factorize the two-dimensional implicit operator to two one-dimensional operators. Accordingly, factorization of Eq. (4.8) yields

\[
\left[ I + \alpha \left( \frac{\partial}{\partial x} [A]^n \right) \right] \left[ I + \alpha \left( \frac{\partial}{\partial y} [B]^n \right) \right] \Delta Q^n - \alpha^2 \frac{\partial}{\partial x} [A]^n \frac{\partial}{\partial x} [B]^n \Delta Q^n = R(Q^n, Q^{n-1})
\]

(4.9)

where \( \alpha = \frac{\theta_1 \Delta t}{1 + \theta_2} \).

In order to achieve this factorization, the quantity \( \alpha^2 \frac{\partial}{\partial x} [A]^n \frac{\partial}{\partial x} [B]^n \Delta Q^n \) was added to the left hand side of Eq. (4.8) and therefore the truncation error (T.E) gets augmented by these terms, but nevertheless, remains of second-order in \( \Delta t \). Although the error does not affect the order of accuracy of the implicit scheme, which is still maintained second-order; it may affect the iterative convergence and stability [Pulliam (1993)]. This error introduced by approximate factorization can be minimized by using sub-iterations, as explained in Section 4.2.3. In Eq. (4.9), each of the two one-dimensional operators forms a block-tridiagonal matrix. Hence, the solution method consists of two one-dimensional sweeps. Each sweep is treated separately by solving the linear system of algebraic equations which have a block-tridiagonal coefficient matrix, using Thomas algorithm. Even though the solution process after approximate factorization is more economical, operations with block tridiagonal matrices still make it computationally expensive. To obtain a solution to this system of equations further efficiently, Pulliam and Chaussee (1981) introduced a diagonalization procedure. This diagonalization method is explained next.
4.1.3. Diagonalization (Pulliam and Chaussee 1981)

The diagonalization procedure developed by Pulliam and Chaussee (1981) uses the eigensystem of the flux Jacobians $[A]^n$ and $[B]^n$ shown in Eq. (4.9) to diagonalize the implicit operators. This is demonstrated by writing

$$A^n = X(Q) \Lambda_A X^{-1}(Q) \ ,$$ \hspace{1cm} (4.10a)

$$B^n = Y(Q) \Lambda_B Y^{-1}(Q) \ ,$$ \hspace{1cm} (4.10b)

where $\Lambda_A$ denotes the diagonal matrix with the eigenvalues of $[A]^n$ as its elements. Similarly, $\Lambda_B$ is the diagonal matrix consisting of the eigenvalues of $[B]^n$, and $X(Q)$ and $Y(Q)$ are the matrices of eigenvectors of $[A]^n$ and $[B]^n$, respectively. Substituting Eq. (4.10a) and (4.10b) in Eq. (4.9) and writing the eigenvector matrices outside the spatial derivatives, we obtain

$$X(Q)\left[I + \alpha \partial_x \Lambda_A\right]X^{-1}(Q)Y(Q)\left[I + \alpha \partial_y \Lambda_B\right]Y^{-1}(Q)\Delta Q^n + O(\Delta t^2) = R(Q^n, Q^{n-1}) \ .$$ \hspace{1cm} (4.11)

The two quantities $X(Q)$ and $Y(Q)$ are related to each other as

$$N(Q) = X^{-1}(Q)Y(Q) \ ,$$

and

$$N^{-1}(Q) = Y^{-1}(Q)X(Q) \ .$$

Substituting Eq. (4.12) in Eq. (4.11),

$$\left\{X(Q)\left[I + \alpha \partial_x \Lambda_A\right]N(Q)\left[I + \alpha \partial_y \Lambda_B\right]Y^{-1}(Q)\right\}\Delta Q^n + O(\Delta t^2) = R(Q^n, Q^{n-1}) \ .$$ \hspace{1cm} (4.12)

This equation represents the diagonalized form of Eq. (4.9). The new implicit operators $\left[I + \alpha \partial_x \Lambda_A\right]$ and $\left[I + \alpha \partial_y \Lambda_B\right]$ are still block tridiagonal, but the blocks are diagonal in
form so that the operators reduce to four independent scalar tridiagonal operators, as per Pulliam and Chaussee (1981). The operations involved in solving now are scalar tridiagonal inversions whereas, prior to diagonalization of Eq. (4.9), the solution process included block tridiagonal inversions. Hence, the computational expense of an implicit scheme reduces significantly with the approximate factorization and diagonalization techniques discussed, thereby improving the efficiency of the time integration technique. Computational experiments by Pulliam and Chaussee (1981) also demonstrated that the steady state solutions before and after diagonalization are identical, as the approximations only affect the implicit side of the scheme. For time accurate simulations, the negative effect of the errors caused by the approximations the implicit side of the scheme can be lessened by using sub-iterations, which will be explained in Sec. 4.2.3. The details of the scheme applied for the spatial-difference operators encountered in the equations so far, is discussed in the following section.

### 4.2. Spatial Discretization

FDL3DI uses higher-order accurate compact-difference schemes for spatial discretization, with tenth-order filtering. To compute the spatial derivative of any discrete quantity \( \phi \) which can be a metric, flux, or flow variable, FDL3DI employs a five-point compact-difference stencil.

In Fig. 4.1, point \( i \) is in the interior of the computational domain. The five-point stencil required to compute the derivative \( \phi' \) at \( i \) is shown in Fig. 4.2. The derivative \( \phi' \) is
calculated by solving the tridiagonal system:

\[
\alpha \phi'_{r+1} + \phi_i' + \alpha \phi'_{r-1} = b \frac{\phi_{r+2} - \phi_{r-2}}{4\Delta \xi} + a \frac{\phi_{r+1} - \phi_{r-1}}{2\Delta \xi}
\]  

(4.13)

where \(\alpha\), \(a\), and \(b\) are constants that determine the spatial properties of the algorithm.

Equation (4.13) is a general formula that includes in it a family of schemes with varying accuracy like standard three-point, second-order accurate explicit scheme (E2), five-point, fourth-order accurate explicit scheme (E4), three-point, fourth-order accurate compact difference scheme (C4), five-point, sixth-order accurate compact difference scheme (C6) etc. The coefficients used for the above mentioned schemes to solve Eq. (4.13) are given in Table 1. Further details are available in Gaitonde and Visbal (1998).

Fig. 4.2. Stencil for first-order derivative in interior of computational domain.

<table>
<thead>
<tr>
<th>Scheme</th>
<th>(\alpha)</th>
<th>(a)</th>
<th>(b)</th>
</tr>
</thead>
<tbody>
<tr>
<td>E2</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>E4</td>
<td>0</td>
<td>4/3</td>
<td>-1/3</td>
</tr>
<tr>
<td>C4</td>
<td>1/4</td>
<td>3/2</td>
<td>0</td>
</tr>
<tr>
<td>C6</td>
<td>1/3</td>
<td>14/9</td>
<td>1/9</td>
</tr>
</tbody>
</table>
When a five-point stencil is considered, special formulations are required for computing the spatial derivatives at the boundary points and at points that are near to the boundary. In Fig. 4.1, 1, 2, M and N represent the points that need special formulation. Equation (4.13) cannot be applied at such points as it utilizes a centered stencil for the approximation of the derivative. Therefore, one-sided stencil is utilized at those points and the required number of points to form the stencil depends on the desired order of accuracy. For higher-order interior schemes for e.g., C6, which stands for sixth-order accurate compact difference scheme, higher-order one-sided formulas may be utilized at boundary points and for points that are near to the boundary. At boundary point 1 shown in Fig. 4.1, the points required to form the one-sided difference stencil are shown in 4.3. The higher-order one-sided difference formula employed at point 1 is presented in Equation (4.13) and the coefficients listed to solve this equation are listed in Table II.

Point 1:  \[ \phi'_1 + \alpha_1 \phi'_2 = a_1 \phi_1 + b_1 \phi_2 + c_1 \phi_3 + d_1 \phi_4 + e_1 \phi_5 + f_1 \phi_6 + g_1 \phi_7 \]  \hspace{1cm} (4.14)

Fig. 4.3. Stencil for first derivative at point 1
Table II. Boundary coefficients for point 1

<table>
<thead>
<tr>
<th>Scheme</th>
<th>$\alpha_1$</th>
<th>$a_1$</th>
<th>$b_1$</th>
<th>$c_1$</th>
<th>$d_1$</th>
<th>$e_1$</th>
<th>$f_1$</th>
<th>$g_1$</th>
<th>OA</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E1$</td>
<td>0</td>
<td>-1</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>$E2$</td>
<td>0</td>
<td>$-\frac{3}{2}$</td>
<td>2</td>
<td>$\frac{1}{2}$</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>2</td>
</tr>
<tr>
<td>$E3$</td>
<td>0</td>
<td>$-\frac{11}{6}$</td>
<td>3</td>
<td>$\frac{3}{2}$</td>
<td>$\frac{1}{3}$</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3</td>
</tr>
<tr>
<td>$E4$</td>
<td>0</td>
<td>$-\frac{25}{12}$</td>
<td>4</td>
<td>$\frac{4}{3}$</td>
<td>$\frac{1}{4}$</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>4</td>
</tr>
<tr>
<td>$E5$</td>
<td>0</td>
<td>$-\frac{187}{50}$</td>
<td>5</td>
<td>$\frac{9}{4}$</td>
<td>$\frac{5}{4}$</td>
<td>$\frac{1}{8}$</td>
<td>0</td>
<td>0</td>
<td>5</td>
</tr>
<tr>
<td>$E6$</td>
<td>0</td>
<td>$-\frac{49}{20}$</td>
<td>6</td>
<td>$\frac{15}{4}$</td>
<td>$\frac{20}{3}$</td>
<td>$\frac{15}{6}$</td>
<td>$\frac{1}{6}$</td>
<td>0</td>
<td>6</td>
</tr>
<tr>
<td>$C2$</td>
<td>1</td>
<td>-2</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>2</td>
</tr>
<tr>
<td>$C3$</td>
<td>2</td>
<td>$-\frac{5}{2}$</td>
<td>2</td>
<td>$\frac{1}{2}$</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3</td>
</tr>
<tr>
<td>$C4$</td>
<td>3</td>
<td>$-\frac{17}{6}$</td>
<td>$\frac{3}{2}$</td>
<td>$\frac{3}{4}$</td>
<td>$\frac{1}{6}$</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>4</td>
</tr>
<tr>
<td>$C5$</td>
<td>4</td>
<td>$\frac{37}{12}$</td>
<td>$\frac{3}{2}$</td>
<td>3</td>
<td>$\frac{2}{3}$</td>
<td>$\frac{1}{12}$</td>
<td>0</td>
<td>0</td>
<td>5</td>
</tr>
<tr>
<td>$C6$</td>
<td>5</td>
<td>$\frac{187}{60}$</td>
<td>$\frac{5}{12}$</td>
<td>5</td>
<td>$\frac{5}{3}$</td>
<td>$\frac{5}{12}$</td>
<td>$\frac{1}{20}$</td>
<td>0</td>
<td>6</td>
</tr>
</tbody>
</table>

Similar to the derivative computation at boundary point 1 explained above, the derivative $\phi'$ at point 2 is also computed. Point 2 is located one point away from the boundary as shown in Fig. 4.1. The points required to form the finite-difference stencil at point 2 is shown in Fig. 4.3 and Eq. (4.15) presents the general formula for the derivative at this point.

Point 2: \[ \alpha_{21}\phi_1' + \phi_2' + \alpha_{22}\phi_3' = a_2\phi_1 + b_2\phi_2 + c_2\phi_3 + d_2\phi_4 + e_2\phi_5 + f_2\phi_6 + g_2\phi_7 \] (4.15)

Fig. 4.4. Stencil for first derivative at point 2

Of the four different kinds of schemes that are possible from Eq. (4.15), coefficients for some schemes (AC4, AC5 and AC6) are listed in Table III. There can be schemes like
BC4, CC4 and DC4 schemes where the first letter in representing the scheme signifies the various options that arise due to the treatment of $\phi'_1$. If the slope at point 1 is treated implicitly, two options arise:

- $\alpha_{21} = \alpha_{22} \neq 0$ referred as option A, LHS of Eq. (4.15) is symmetric about point 2 and the coefficients so obtained are shown in Table III,
- $\alpha_{21} \neq \alpha_{22} \neq 0$ referred as option B, maintaining the same stencil as in option A, the order of accuracy is increased by one,

Two other options arise from the decoupling of the slopes at the endpoints from the rest of the domain by setting $\alpha_{21} = 0$. They are:

- $\alpha_{21} = 0, \alpha_{22} \neq 0$ referred as option C, the coefficients are computed implicitly and
- $\alpha_{21} = 0, \alpha_{22} = 0$ referred as option D, complete explicit formulation is used.

### Table III. Boundary coefficients for point 2

<table>
<thead>
<tr>
<th>Scheme</th>
<th>$\alpha_{21}$</th>
<th>$\alpha_{22}$</th>
<th>$a_2$</th>
<th>$b_2$</th>
<th>$c_2$</th>
<th>$d_2$</th>
<th>$e_2$</th>
<th>$f_2$</th>
<th>$g_2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>AC4</td>
<td>$\frac{1}{4}$</td>
<td>$\frac{1}{4}$</td>
<td>$-\frac{3}{4}$</td>
<td>0</td>
<td>$\frac{3}{4}$</td>
<td>0</td>
<td>$\frac{3}{4}$</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>AC5</td>
<td>$\frac{3}{14}$</td>
<td>$\frac{3}{14}$</td>
<td>$-\frac{15}{28}$</td>
<td>$-\frac{5}{28}$</td>
<td>$\frac{6}{39}$</td>
<td>$-\frac{1}{14}$</td>
<td>$\frac{1}{84}$</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>AC6</td>
<td>$\frac{1}{11}$</td>
<td>$\frac{2}{11}$</td>
<td>$-\frac{30}{33}$</td>
<td>$-\frac{30}{119}$</td>
<td>$\frac{34}{39}$</td>
<td>$-\frac{7}{39}$</td>
<td>$\frac{5}{39}$</td>
<td>$-\frac{1}{132}$</td>
<td>0</td>
</tr>
</tbody>
</table>

So far, computing the first derivative at points 1 and 2 as shown in Fig. 4.1 is discussed. To compute first derivative at points N and M (=N-1) shown in Fig. 4.1, the coefficients in Table II and Table III can be applied by changing the sign of the coefficients.

Point M:
\[
\alpha_{M1} \phi'_{N-2} + \phi'_{N-1} + \alpha_{M2} \phi'_{N} = a_M \phi_N + b_M \phi_{N-1} + c_M \phi_{N-2} + d_M \phi_{N-3} + e_M \phi_{N-4} + f_M \phi_{N-5} + g_M \phi_{N-6}
\] (4.16)

Point M:

\[
\alpha_{M1} \phi'_{N-2} + \phi'_{N-1} + \alpha_{M2} \phi'_{N} = a_M \phi_N + b_M \phi_{N-1} + c_M \phi_{N-2} + d_M \phi_{N-3} + e_M \phi_{N-4} + f_M \phi_{N-5} + g_M \phi_{N-6}
\] (4.17)

Point N:

Up till now, the technique to compute first-order derivative of \( \phi \) such as a metric, flux, or flow variable, for a range of accuracies with a maximum of sixth-order accuracy has been discussed. Similarly, the inviscid fluxes are computed in transformed coordinates at every node and are differentiated further using Eq. (4.11). For the viscous fluxes, the primitive variables \( u, v, w \) and \( T \) are first differentiated, and the stress tensor is formed at each node. The viscous terms are then computed by another application of the Eq. (4.11).

The higher-order compact difference schemes used in the present study formulated by Lele (1992), utilizes central-difference for the approximation of spatial
derivatives. Centered schemes are non-dissipative in nature, which makes these schemes susceptible to numerical instabilities. Numerical instabilities might arise due to inadequate mesh refinement in the regions of large gradients, explicit boundary conditions etc. It is necessary to suppress the growth of these numerical instabilities or non-physical oscillations in order to obtain a reliable solution; otherwise, it may destroy the fidelity of the solution. For central-difference schemes, high frequency oscillations can be suppressed by adding smoothing (artificial viscosity). In order to extend the flow solver to more practical simulations, while retaining the improved accuracy of the spatial compact discretization, a high-order implicit filtering technique is employed [Visbal and Gaitonde (2002)]. The filters employed in this work are derived by Gaitonde et al. (1997) to stabilize finite-volume schemes in electromagnetic wave phenomena. The details of the order of accuracy of the scheme, the computational stencil involved and the role of control parameter for the filtering technique are explained in the following section.

4.3. Filtering

A non-dispersive spatial filter (up to tenth-order) is incorporated in the methodology for reasons stated at the end of Sec. 4.2.2. The filtered values \( \hat{\phi} \) of the desired variable \( \phi \) are obtained by solving a tridiagonal system of equations,

\[
\alpha_f \hat{\phi}_{i-1} + \hat{\phi}_i + \alpha_f \hat{\phi}_{i+1} = \sum_{n=0}^{5} \frac{a_n}{2} (\phi_{i+n} + \phi_{i-n}) ,
\]

and the computed (updated) value \( \phi \) is replaced with \( \hat{\phi} \). For e.g., if the variable that needs to be filtered is the solution vector \( Q \), then the tridiagonal system is
\[ \alpha_f Q_{i-1} + Q_i + \alpha_f Q_{i+1} = \sum_{n=0}^{\frac{\pi}{2}} \frac{a_n}{2} (Q_{i+n} + Q_{i-n}) \quad (4.19) \]

In the present study, the variable that is filtered is the solution vector \( Q \) and the filter is applied to the variable in each of the three computational directions, after each sub-iteration. In Eq. (4.19), \( \alpha_f \) is a free parameter that controls the degree of filtering. \( \alpha_f \) lies in the range \(-0.5 \leq \alpha_f \leq 0.5\) depending on the spectral function (or frequency response); detailed explanation is available in the technical report by Gaitonde and Visbal (1998). Higher the order of accuracy of the filter and higher the value of \( \alpha_f \), lower is the dissipative nature of the filter. The coefficients for the different orders of filter (up to tenth-order), \( a_n \), are listed in Table IV [Gaitonde and Visbal (1998)].

### Table IV. Coefficients for filter formula at interior points

<table>
<thead>
<tr>
<th>Scheme</th>
<th>( a_0 )</th>
<th>( a_1 )</th>
<th>( a_2 )</th>
<th>( a_3 )</th>
<th>( a_4 )</th>
<th>( a_5 )</th>
<th>OA</th>
</tr>
</thead>
<tbody>
<tr>
<td>( F_2 )</td>
<td>( \frac{1}{2} + \alpha_f )</td>
<td>( \frac{1}{2} + \alpha_f )</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>2</td>
</tr>
<tr>
<td>( F_4 )</td>
<td>( \frac{5}{8} + \frac{3\alpha_f}{4} )</td>
<td>( \frac{1}{2} + \alpha_f )</td>
<td>( -\frac{1}{8} + \frac{\alpha_f}{4} )</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>4</td>
</tr>
<tr>
<td>( F_6 )</td>
<td>( \frac{11}{16} + \frac{5\alpha_f}{8} )</td>
<td>( \frac{15}{32} + \frac{17\alpha_f}{16} )</td>
<td>( -\frac{3}{16} + \frac{3\alpha_f}{8} )</td>
<td>( \frac{1}{32} - \frac{\alpha_f}{16} )</td>
<td>0</td>
<td>0</td>
<td>6</td>
</tr>
<tr>
<td>( F_8 )</td>
<td>( \frac{93 + 70\alpha_f}{128} )</td>
<td>( \frac{7 + 18\alpha_f}{16} )</td>
<td>( -\frac{7 + 14\alpha_f}{32} )</td>
<td>( \frac{1}{16} - \frac{\alpha_f}{8} )</td>
<td>( -\frac{1}{128} + \frac{\alpha_f}{64} )</td>
<td>0</td>
<td>8</td>
</tr>
<tr>
<td>( F_{10} )</td>
<td>( \frac{195 + 126\alpha_f}{256} )</td>
<td>( \frac{105 + 302\alpha_f}{256} )</td>
<td>( \frac{15(-1 + 2\alpha_f)}{64} )</td>
<td>( \frac{45(1 - 2\alpha_f)}{128} )</td>
<td>( \frac{5(-1 + 2\alpha_f)}{256} )</td>
<td>( 1 - 2\alpha_f )</td>
<td>10</td>
</tr>
</tbody>
</table>

With proper choice of coefficients, Eq. (4.19) provides a \( 2N \)th order formula for a \( 2N + 1 \) point computational stencil. Special formulas are required at near boundary points, where Eq. (4.19) cannot be applied. Two approaches are suitable for the treatment of variables at these points. Firstly, the order of accuracy is reduced upon approaching the boundary till a centered stencil is available as proposed by Visbal and Gaitonde (1999). Absolute accuracy is retained by optimizing the value of \( \alpha_f \); values
of $\alpha_f$ between 0.3 and 0.5 are mostly applicable. As second approach, higher-order one-sided formulas can be used. Unlike the interior filter, the one-sided high-order filters can amplify wave amplitudes for certain range of wave numbers. To eliminate the undesirable amplification behavior, higher values of $\alpha_f$ may be used near boundaries. In the present work, the first approach is followed. To describe the discretization and filter used for any flow simulation, the order of the interior filtering formula is appended to the scheme designation. For example, C6F10 represents sixth-order compact scheme combined with a tenth-order filter. Near the boundaries, fourth-order one-sided filter is employed at the first point away from the domain boundary, a sixth-order filter at the second point away, and an eighth-order filter at the third and fourth points away from the end of the domain boundary.

Low-pass filtering provides dissipation at the high modified wave numbers only when the spatial discretization already exhibits significant dispersion errors, whereas upwind biased schemes introduce dissipation across a wide range of wave numbers. This warrants the incorporation of filtering technique in conjunction with high-order compact difference schemes.

4.4. Method of solution

To reduce errors due to linearization, factorization and diagonalization discussed in Sec. 4.2.1 and 4.2.2, Newton-like sub-iterations are incorporated in the flow solver. Sub-iterations improve the temporal accuracy and stability properties of the algorithm. With the application of implicit three-point backward discretization for temporal derivatives, the approximately-factored equation in the delta form for the sub-iterative step is written as
\[ I + \left( \frac{2\Delta t}{3} \right) \delta_{e^2} \left( \frac{\partial F^p}{\partial Q} - \frac{1}{Re} \frac{\partial F^p_v}{\partial Q} \right) \times I + \left( \frac{2\Delta t}{3} \right) \delta_{\eta^2} \left( \frac{\partial G^p}{\partial Q} - \frac{1}{Re} \frac{\partial G^p_v}{\partial Q} \right) \times \]
\[ I + \left( \frac{2\Delta t}{3} \right) \delta_{\xi^2} \left( \frac{\partial H^p}{\partial Q} - \frac{1}{Re} \frac{\partial H^p_v}{\partial Q} \right) \Delta Q \]
\[ = -\left( \frac{2\Delta t}{3} \right) \left( \frac{1}{2\Delta t} \right) \left( 3Q^p - 4Q^n + Q^{n-1} \right) + \delta_{\xi^2} \left( F^p - \frac{1}{Re} F^p_v \right) + \delta_{\eta^2} \left( G^p - \frac{1}{Re} G^p_v \right) \]
\[ + \delta_{\xi^2} \left( H^p - \frac{1}{Re} H^p_v \right), \quad (4.20) \]

where \( \Delta Q = Q^{p+1} - Q^p \), and \( p \) denotes the sub-iteration number. For the first sub-iteration (\( p = 1 \)), \( Q^p = Q^n \), and with subsequent sub-iterations, \( Q^{p+1} \to Q^{p+1} \). The spatial derivatives in the implicit operators are represented using standard second-order centered approximations whereas high-order discretizations are employed for the residual. Although not shown in Eq. (4.20), nonlinear artificial dissipation terms are appended to the implicit operator (only) to enhance stability. These terms include both second and fourth-order dissipation operators. The diagonal form of the implicit algorithm then gives rise to a scalar pentadiagonal system. Sub-iterations also eliminate the impact of the implicit damping coefficients on the final solution [Visbal and Gaitonde (2002)] and therefore the choice of these coefficients is only from the stability perspective.

The details of the numerical scheme used for the approximation of spatial and temporal derivatives in FDL3DI, were described in this chapter. Also, the solution procedure utilized in the flow solver is explained. In the next chapter, results obtained will be discussed in detail.
Chapter 5

Results and Discussion

This chapter presents the results obtained from the numerical study of flow separation control for cylinder flow and LPT cascade flow. Results for two additional configurations are also included. First, in order to validate the simulation of a turbulent wake, a model problem of flow over a circular cylinder is studied at $Re = 3,900$, and the results obtained are compared with the available experimental and numerical results. Then, the simulation is performed for $Re = 13,400$, and preliminary baseline results are obtained. The first section describes the details, and discusses the results of the model problem of flow past a circular cylinder at $Re_D = 3,900$, which is considered as a validation study for computing the separated flow using the flow solver FDL3DI. The second section includes the details of the computation, and discusses the baseline results obtained for cylinder flow at $Re = 13,400$ on a relatively coarser grid. This case is a first step towards studying the implementation of flow control techniques for preventing or delaying separation. Comparison of the results for $Re_D = 3,900$ and $Re_D = 13,400$ is also discussed in this section. The third section presents the effect of the grid density on the evolving flow solution for the cylinder flow at $Re_D = 13,400$. The fourth section describes in detail the implementation of control, using VGJs for the cylinder flow at $Re_D = 13,400$, followed by the discussion of the results for flow separation control. The final section deals with the baseline solution for LPT ‘PAK-B’ blade geometry at $Re = 25,000$ followed by the discussion of results for flow separation control using VGJs.

A fourth-order accurate compact-difference scheme along with sixth-order filtering (C4F6) has been used for the present numerical study, unless explicitly
mentioned about the order of accuracy. For all of the simulations, a single or multi
topology grid has been used, based on the requirements of the geometry. A multi-block
structured grid approach has been selected to benefit from the parallelized flow solver, to
optimize the computational resources to a greater extent, and reduce the required turn-
around time for a simulation. PEGSUS, a NASA Ames research code, is used for
determining the connectivity information at the block interfaces for the multi-block
structured grids.

5.1 Circular Cylinder at Re = 3900

Flow past a circular cylinder at Re_D = 3,900 and M_∞ = 0.2 has been studied as a
validation test case for the correct usage of the parallelized LES version of the FDL3DI
flow solver. In this study, a multi-block, structured grid with O-topology is used as shown
in Fig. 1. For this flow configuration, the far-field boundary is located 75 diameters away
from the center of the cylinder. The diameter D of the cylinder is 1.0 unit, and the
spanwise extent of the cylinder is \( \frac{\pi D}{2} \). An O-grid, is employed, with 100 points in the
circumferential direction, 100 points in the radial direction, and 30 points in the span-
wise direction. The grid points are spaced uniformly in the circumferential and spanwise
direction, and are clustered in the radial direction, using a hyperbolic tangent function,
with the first grid-point non-dimensional spacing equal to 7.7 x 10^{-5}. To use the parallel
flow solver, the domain is decomposed into 8 blocks. The partitions are made only in the
circumferential direction. Each block is extended into adjacent blocks, such that an
overlap of four cells was established at each block interface. Because all of the
overlapping mesh points in this decomposition are coincident, no interpolation is
required. PEGSUS, a NASA Ames research code, is used to identify donor and recipient grid points in the overlapping domains. Each block is then assigned to a separate processor and, among the processors, the flow data is updated for every iteration at the domain boundaries, using the standard Message Passing Interface (MPI) send and receive library calls.

Boundary conditions are prescribed on the closed domain such that free-stream conditions prevail along the upstream boundary, and all dependent variables have zero first-order derivatives across the downstream boundary. On the surface of the cylinder, Pressure Neumann boundary conditions are specified and an isothermal wall is specified with no-slip conditions are applied on the surface of the cylinder. Periodicity is used in the span-wise direction. The initial conditions correspond to that of uniform flow. A constant non-dimensional time step $\Delta t = 0.001$ is used for the entire simulation, along with 2 sub-iterations per global time step. The simulation is carried out for up to 200 non-dimensional time units. The mean quantities are gathered by time-averaging the solution over 50 characteristic times, and by averaging along the span.

The mean flow statistics from the current simulations for $Re = 3,900$ are plotted in Figs. 2-5, together with results from previous studies for comparison. The mean $C_p$ on the surface of the cylinder is plotted in Fig. 2, and is compared with the experimental results of Norberg (1994) for approximately the same Re (4200). The maximum value for pressure occurs at the stagnation point ($\theta = 0^0$) and thereafter, the pressure decreases till about $\theta = 70^0$. The flow separates at about $\theta = 80^0$. 
The present simulation predicts a smaller recirculation zone behind the cylinder than the B-spline simulations by Kravchenko and Moin (1998), as can be seen in Fig. 3. It is observed that for the present simulation the length of the recirculation zone ($L_r/D$) is 1.20, while for the B-spline simulation is 1.35. The reason for this could be the premature transition predicted by the coarse grid employed for the current numerical computations, thereby affecting the extent of the recirculation region.

The mean streamwise velocity, $\bar{u}$, and the mean crossflow velocity $\bar{v}$ are extracted for three streamwise locations, and shown in Fig. 4 and Fig. 5, respectively, in the very near-wake region (up to $x/D = 3.0$), and are compared with the B-spline simulation of Kravchenko and Moin (1998). It can be observed that, away from the cylinder, $\bar{u}$ approaches the value $U_\infty$, and $\bar{v}$ approaches zero, in the absence of any freestream turbulence. The results agree well, but not exactly, with those of the earlier published results. Despite the fact that a fourth-order finite-differencing scheme is used, the small differences can be attributed to the coarseness of the grid which leads to high truncation errors. Also, at $x/D = 1.06$, the B-spline simulations show a U-shaped profile for $\bar{u}$, while the current simulation displays a V-shape trend, which is actually expected further downstream. It is believed that the grid employed in the present computation predicted an earlier transition, resulting in the higher value of the fluctuations close to the cylinder and, hence, led to a V-shaped behavior.

The instantaneous contours of z-vorticity are shown in Fig. 6. It can be observed that boundary layer separates from the upper and lower sides of the cylinder, and rolls up, forming clockwise and counterclockwise vortices, depicted in blue and red colors, respectively, which subsequently are convected alternately into the wake. The pattern
formed because of vortex shedding like this is known as Karman vortex street. The wake consists of a wide range of length scales, and is highly unsteady and three-dimensional in nature. Figure 7(a) shows the contours of spanwise Reynolds stress, $w^2$. This quantity is an indicator of the three-dimensionality in the flow. The result indicates that the maximum occurs somewhere downstream in the wake region where the vortices from the upper and lower surfaces interact with each other. The region with significant three-dimensionality extends up to $x/D = 6$. The turbulence kinetic energy plot in Fig. 7(b) shows a variation which is similar to that observed for the spanwise fluctuation, with peak values confined in the wake region after separation.

In order to examine the dynamics of the separated flow in the wake region, the temporal variation of the streamwise velocity ($u$) has been examined. Three points where data has been extracted are shown in Fig. 8(a). In order to extract the true dynamics of the wake, Point 2 and Point 3 are ensured to lie in the path of the vortex structures. Figure 8(b) shows the temporal variation of $u$, and it can be seen that the variation at all the locations does not exhibit any obvious periodic component. This reflects the truly unsteady nature of the flow in the wake region.

5.2. Circular Cylinder at $Re = 13,400$

Having validated the procedure, results are obtained next for $Re = 13,400$. The grid, boundary conditions and initial conditions used for this simulation are same as those used in the simulation at $Re_D = 3,900$. Instantaneous contours of vorticity magnitude are shown in Fig. 9. Two shear layers separating from the upper and lower surfaces of the cylinder and formation of the Karman vortex street are seen in the figure. From the contour plot of spanwise Reynolds stress $w^2$ in Fig. 10, it is observed that the value of
\( \bar{w}^2 \) at \( \text{Re} = 13,400 \) is significantly greater than the value at \( \text{Re} = 3,900 \), implying that the flow at \( \text{Re} = 13,400 \) is highly three-dimensional as compared to the flow at \( \text{Re} = 3900 \). Figure 11 shows the contour plot of the turbulence kinetic energy, and the maximum from the contour legend is 0.34 as compared to the maximum value of 0.31 for \( \text{Re} = 3,900 \), implying that the flow is more turbulent for \( \text{Re} = 13,400 \). Profiles of \( \bar{u} \) and \( \bar{v} \) are shown in Fig. 12. Qualitatively, the trend of these profiles is similar to that for \( \text{Re} = 3900 \). It should be pointed out that the streamwise velocity at \( x/D = 1.06 \) displays a v-shaped nature. It is hypothesized that the reason for this could be because of the coarseness of the grid used, thereby leading to premature transition to turbulence.

Contours of instantaneous streamwise, crossflow and spanwise velocities in the region occupying the first seven diameters downstream of the cylinder are presented in Fig. 13. Figures 13(a) and Fig. 13(b) show the contours of streamwise and spanwise velocity, respectively, while Fig. 13(c) presents the contours of crossflow velocity. Figure 13(c) clearly shows the alternating regions of positive and negative crossflow velocity corresponding to the Karman vortices. The wake at this Re is comprised of both small and large structures and, as the streamwise distance increases, the flow structures tend to increase in size, as can be seen in Fig. 13(b), which represents contours of spanwise velocity. However, it should be pointed out that as the streamwise distance increases, the grid opens up rapidly, and is not suitable for capturing the smaller structures, which are expected to exist in the wake at this Re. Figure 14 shows the comparison of the distribution of the coefficient of pressure (Cp) at the surface of the cylinder at \( \text{Re} = 13,400 \) with the experimental study by Sarioglu and Yavuz (2002) at \( \text{Re} = 16,024 \), and the Cp surface distribution is found to match reasonably well within the experimental
accuracy limits. Figure 15 shows the time- and spanwise averaged streamwise velocity at
the centerline in the wake region and it is observed that the length of the recirculation
region, \((L_r/D)\) is 1.06 as compared to the \(L_r/D = 1.20\) for \(Re = 3,900\).

5.3 **Circular Cylinder at \(Re = 13400\) with grid refinement**

To conduct a grid-independence study, the 3-D simulation for \(Re = 13,400\) is also
performed on a fine grid, which will be referred to as Grid-2 in the remainder of the
discussion. A snapshot of the grid-2 used for this simulation is shown in Fig. 16. The
grid consists of coarse- and fine-grid regions, and is designed in such a way that
appropriate resolution is achieved in the computational domain. The set-up of this grid
reduces the total number of grid points, saving CPU time and memory. It contains a total
of 3.9 million grid points, with 48 points in the spanwise direction. Approximately, 70% of
the total grid points are concentrated in the wake region, to capture the smaller scales
which are expected to exist in the wake region at this \(Re\). The first near-wall non-
dimensional spacing is \(6.9 \times 10^{-5}\). The surface of the cylinder is located at \(r/D = 0.5\),
whereas the inflow and outflow boundaries are located at \(r/D = 30\). The grid is
decomposed into 24 blocks and, in order to reduce the computation time the initialization
for the solution on grid-2 is taken from the coarse-grid solution. The simulation is
currently ongoing and only a small time, \(t = 15T\), instantaneous results are presented
here.

5.3.1 **Vorticity Contours**

The calculated z-vorticity for Grid-2 is presented in Fig. 17. Figure 18 shows the
comparison of the z-vorticity contours, with the grid overlaid on the contours, for the
coarse grid and grid-2. It can be easily observed that grid-2 has captured the smaller
scales which were not seen in the coarse-grid simulation and are expected to exist in the wake region at this Re. Also, Grid-2 captures the structures farther downstream which the coarse-grid was not able to. The formation of the Karman vortex street is clearly visible in the simulations performed on Grid-2, as shown in Fig. 19, while in the coarse-grid simulation, this phenomena was not vividly seen.

5.3.2 Time-history of lift and drag coefficient

Time-history of lift coefficient is calculated and is presented in Fig. 20. It is clear from Fig. 20 that the lift coefficient displays a repeated pattern with time and the time-period of the pattern is about 5000 iteration, which represents 5 characteristic times, yielding a Strouhl number of about 0.2, for this flow, which is same as what was observed in the experiments Borgeson (2002).

5.3.3 Q-criterion

Three views of instantaneous isosurfaces of the Q-criterion are calculated and are shown in Fig. 21. The isosurfaces corresponds to Q = 4. The Q-criterion helps to display the coherent vortical flow structures Dubeif and Delcayre (2000). Q is defined as,

\[ Q = \frac{1}{2} \left( \Omega_x \Omega_y - S_x S_y \right) \]

and is also equal to:

\[ Q = \frac{\nabla^2 p}{2 \rho} \]

Q is calculated over the whole flow field using the equations in Techplot9.0. The figure clearly shows that the flow transitions to turbulence just after the shear layer separates from the surface of the cylinder, which is the typical feature of the flow at this Re. The
presence of 3-D turbulent flow structures as well as the development of Karman vortex street is vividly seen in the wake region.

5.3.4 Wake Profiles Comparison for Fine and Coarse Grids

The effect of grid-refinement on the profiles of streamwise and crossflow velocities at three locations in the wake region is shown in Figs. 22-23. So as to facilitate comparison, the three locations in the wake, where the profiles are plotted, are the same as used in the coarse-grid simulation. From the comparison of streamwise velocity, it can be clearly concluded that the V-shaped nature of the streamwise velocity profile, at $x/D = 1.06$ is because of the coarseness of the grid employed, as was hypothesized in Section 5.2.

5.4 Flow Separation Control for Circular Cylinder at $Re = 13,400$

A separate case is run, keeping all the other parameters same as the case discussed in the previous section, but with spanwise distance of 0.15, which can be considered as a baseline case for the flow separation control application. The cylinder case takes 20 days running time for the whole simulation on four million grid points. The reason for choosing the smaller spanwise distance is to provide an adequate resolution for VGJ holes with the same number of points in the $z$-direction, as the case discussed in previous section. This would allow the simulation to finish in a reasonable amount of time.

After obtaining the results for the baseline simulation with spanwise distance of 0.15, it is observed from the time-averaged vorticity contours, that the time-averaged flow separates at a point roughly just before $\theta = 90^0$, and with this understanding, the VGJs are placed just before the separation point, at $\theta = 75^0$. The grid used for this simulation is the same as that for the baseline simulation with $\Delta z = 0.15$. The
initialization for the simulations presented in this section is taken from the solution obtained from the case with spanwise distance of $\frac{\pi D}{2}$ and, for meaningful comparison between the baseline and control cases, the same initialization is used for both the cases. The simulation is carried out for 42 characteristic times after initialization. The VGJs are turned on after 25 characteristic times following initialization, and the time-averaging is started at 30 characteristic times after initialization. The following section describes in detail the boundary conditions and parameters that were used in the separation control study for the cylinder flow.

5.4.1 VGJ Hole Geometry and Parameters

In the present simulation, the VGJ hole is approximated using a square shaped geometry. The width of the hole in the spanwise direction is 3 grid cells and width in circumferential direction is 4 grid cells. The hole location and size are shown in Fig. 24. A pitch angle ($\beta$) of 30 degree and a skew angle ($\gamma$) of $90^0$ is considered in the present simulation. The pitch angle is defined as the angle that the jet makes with its projection on the local surface, and the skew angle is the angle which projection of the jet makes with the local freestream direction. The jet exit velocity vector consists of components in the x, y and z directions, and these are defined in the following paragraphs.

VGJs are placed at $\phi = 75^0$

The component of the jet velocity normal to the cylinder’s surface is calculated as:

$$V_r = V_{\text{max}} \sin \beta$$

The component of the jet velocity along the surface parallel to surface is calculated as:

$$V_\theta = V_{\text{max}} \cos \beta \cos \gamma$$
Component of the jet velocity along the $z$-direction is calculated as:

$$V_z = V_{\max} \cos \beta \sin \gamma$$

$$U = V_0 \sin \phi - V_r \cos \phi$$

$$V = V_0 \cos \phi + V_r \sin \phi$$

$$U = V_{\max} \left( \cos \beta \cos \gamma \sin \phi - \sin \beta \cos \phi \right)$$

$$V = V_{\max} \left( \cos \beta \cos \gamma \cos \phi + \sin \beta \sin \phi \right)$$

Jet hole is assumed to be isothermal while the pressure over the jet holes is calculated by solving the inviscid normal-momentum equation, and all the derivatives are approximated using a fourth-order accurate expression.

$$-\frac{\partial p}{\partial r} = V_r \frac{\partial V_z}{\partial r} - \frac{V_0^2}{r}$$

$$\frac{\partial p}{\partial r} = \frac{V_0^2}{r} - V_r \frac{\partial V_r}{\partial r}$$

To approximate $\frac{\partial V_z}{\partial r}$ and $\frac{\partial p}{\partial r}$ a fourth-order accurate one-sided expression is used

$$\frac{\partial V_z}{\partial r} = -\frac{25}{12} V_{r, 1, JS, K} + 4 V_{r, l, JS+1, K} - 3 V_{r, 1, JS+2, K} + \frac{4}{3} V_{r, 1, JS+3, K} - \frac{1}{4} V_{r, 1, JS+4, K}$$

$$\frac{\partial p}{\partial r} = -\frac{25}{12} p_{l, JS, K} + 4 p_{l, JS+1, K} - 3 p_{l, JS+2, K} + \frac{4}{3} p_{l, JS+3, K} - \frac{1}{4} p_{l, JS+4, K}$$
Since the value of skew angle used is $90^0$, $\theta = 0$.

$$\Rightarrow \frac{-25}{12} p_{l, JS, K} + 4p_{l, JS+1, K} - 3p_{l, JS+2, K} + \frac{4}{3} p_{l, JS+3, K} - \frac{1}{4} p_{l, JS+4, K} = \frac{V_r^2}{r} - V_r \frac{\partial V_r}{\partial r}$$

$$\Rightarrow p_{l, JS, K} = \frac{12}{25} \left( 4p_{l, JS+1, K} - 3p_{l, JS+2, K} + \frac{4}{3} p_{l, JS+3, K} - \frac{1}{4} p_{l, JS+4, K} + V_r \frac{\partial V_r}{\partial r} \right)$$

In the present simulation, a uniform jet velocity is provided across the hole with a blowing ratio of 2.0. The jets were pulsed with a non-dimensional frequency ($F^+$) of 1.0. The time-period of the jet corresponding to $\Delta t = 0.001$ is 1.0, which is equivalent to 1000 time steps. The jets were issued with duty cycle ratio of 50%, which means that the jets were active for first 500 time steps, and inactive for other 500 time steps.

### 5.4.2 Lift and Drag Coefficient Time-Histories

Time-histories of lift and drag coefficients are obtained for baseline and control cases and are overlaid in the same plot to facilitate comparison, as shown in Fig. 25 and Fig. 26. In an unsteady separated flow, the location of the separation point oscillates, and it becomes extremely difficult to assess the flow control strategy. In these situations, lift and drag prove to be very useful quantities for monitoring efficacy of the flow control strategy. The lift and drag time-histories are collected from the time when the VGJs are turned on till the end of the simulation. The VGJs are turned on at $t/\tau = 25$, and the time-
averaging is started at $\tau = 30$. The statistics are then collected till $\tau = 46$, that is over 16 characteristic times. It can be easily seen, in Fig. 25, that the drag reduces significantly after the VGJs are turned on.

The drag force acting on the cylinder is comprised of the pressure force and the viscous force. In order to investigate the reason for the reduction in the drag coefficient, the drag coefficient time-history due the pressure forces alone due to the viscous forces alone is calculated, and is shown in Fig. 27. As is expected and can be concluded from this figure, force due to pressure for flow past a circular cylinder has a larger share in the total drag acting on the cylinder. Also, both viscous as well as pressure drag decrease after the VGJs are turned on, but since pressure drag is significantly greater than viscous drag, it can be said that the reduction in total drag is solely because of reduction in pressure drag.

5.4.3 Cp Variation along the Surface of the Cylinder

The variation of Cp along the surface of the cylinder for controlled case is compared with baseline Cp distribution, and is shown in Fig. 28. The VGJs are applied on the upper half of the cylinder, and therefore, the Cp distribution plotted only for the upper half. The effect of the VGJs is to decrease the pressure upstream of the cylinder’s shoulder while increasing it downstream of the cylinder’s shoulder. Because of the reduced pressure in the upstream portion, the force due to pressure in the forward direction reduces, leading to the reduction in drag. Similarly, the pressure increase in the downstream portion increases the force due to pressure in backward direction, and consequently, contributes to drag reduction.
5.4.4 Animation of Vorticity contours

Animation of instantaneous z-vorticity contours is created at the mid z-section for the baseline and the control cases. From the animation, it is clear that the shear layer separating from the upper surface (where the VGJs are implemented) transitions to turbulent earlier than the shear layer separating from the lower surface of the cylinder. Also, it can be seen that the vertical extent of the wake has been reduced by the application of VGJs.

5.4.5 Momentum thickness and Total pressure

The wake region behind the cylinder is also considerably reduced by flow control. This reduction may be quantified by the integrated wake pressure loss coefficient $C_w$ defined as:

$$C_w = \int \left( \frac{P_n - P_{i0}}{P_n - P_i} \right) dy$$

where the integration is performed across the wake profile between the boundaries in the vertical direction. The total pressure is plotted in the wake region at 3 locations ($x/D = 3.0$, $x/D = 5.0$ and $x/D = 7.0$), as shown in Figs. 29-31. From Fig. 29, it is clear that application of flow control has led to higher total pressure in the wake region. It is found that $C_w = 2.16$ for the baseline flow, and $C_w = 1.95$ for the controlled flow, this represents approximately 10% reduction in loss coefficient. Similar effect of control can be observed at $x/D = 5.0$ and $x/D = 7.0$. It represents about 9% and 8% reduction in loss coefficient at $x/D = 5.0$ and $x/D = 7.0$ respectively.

The profiles for $U_{avg} (1 - U_{avg})$ are shown in Fig. 32. To calculate the momentum thickness, the quantity, $U_{avg} (1 - U_{avg})$ is integrated in the vertical direction, in the wake
region, at x/D = 5.0. Upon integrating the quantity mentioned above, the momentum thickness is found to reduce by 30% after the application of control.

5.4.6 Efficiency of Control

For energy considerations, let us rephrase the statement of the problem in a slightly different way, by saying that a cylinder is pulled through a viscous fluid with a time-varying force $F$ such that it translates with a constant velocity $U_\infty$, and experiences a time-varying drag force $D$. Since the cylinder is translating at constant velocity at any instant, the magnitude of $F$ must be equal to the magnitude of $D$. For this, the external work per unit time required to be done by the force $F$ at any time $t = t_o$ must be equal to

$$ dW = F(t_o) U_\infty. $$

Then, work done by the force $F$ during the time interval, $t = t_1$ to $t = t_2$ can be calculated as

$$ W = U_\infty \sum_{t_1}^{t_2} F \Delta t. $$

For the baseline case,

$$ W_b = U_\infty \sum_{t_1}^{t_2} F_b \Delta t $$

For the controlled case,

$$ W_c = U_\infty \sum_{t_1}^{t_2} F_c \Delta t $$

The net energy saved because of drag reduction by VGJs is

$$ E_s = U_\infty \sum_{t_1}^{t_2} (F_b - F_c) \Delta t $$
The value of $E_s$ can be readily calculated using the $C_D$ time history. The energy induced ($E_i$) by the VGJs in the flow at any instant, provided the jet is active at that instant, can be calculated as explained in the following paragraph.

$$\frac{dE_i}{dt} = \frac{1}{2} m (2U_\infty)^2 = \frac{1}{2} (2 \rho_\infty W L U_\infty) (2U_\infty)^2$$

Then, the energy added to the flow from $t = t_1$ to $t = t_2$ is:

$$E_i = \sum_{t_1}^{t_2} \frac{1}{2} (2 \rho_\infty W L U_\infty) (2U_\infty)^2 \Delta t$$

$$E_i = \frac{1}{2} (2 \rho_\infty W L U_\infty) (2U_\infty)^2 (t_2 - t_1)$$

For the present analysis, $t_1 = 28.0$ and $t_2 = 34.0$ is chosen.

$E_i = 0.0074$

$E_s = 1.6901$

This means that 0.5 units of energy supplied to the flow helps in saving 100 units of energy.

**5.5 Separation flow control results for LPT at Re = 25,000**

This section details the preliminary results obtained for the flow separation control study of three-dimensional flow through a low-pressure turbine cascade. A 3D flow through a low-pressure turbine cascade with and without separation control strategy has been investigated in this study. For this purpose, a Pratt & Whitney PAK-B low-pressure blade is used in a linear cascade arrangement. Figure 33 shows the flow configuration, with this blade profile. All the results presented in this section are obtained for Reynolds number of 25,000 (based on the inlet velocity and axial chord), and a reference Mach number of 0.1. A 12-block structured grid (Fig. 34), generated by
Mutnuri (2003) using the grid-generation software, GridPro, is used for the present analysis. The grid consists of 2 million grid points, with 32 planes along the spanwise direction. The upstream boundary for this grid is located at a distance of 1.5 $C_{ax}$ from the leading edge of the blade, and the downstream boundary is located at a distance of 5 $C_{ax}$ with spanwise extent of the domain equal to 0.15 $C_{ax}$. A simulation for the baseline case is performed first. This simulation will serve as the baseline solution for the control case. The reason for choosing spanwise extent of only 0.15 $C_{ax}$ is to provide the adequate resolution for the VGJ hole, as was explained in Section 5.4. A constant time step of $\Delta t = 0.0005$ is used to obtain the results for the present simulation. The solution is advanced in time using second-order temporal accuracy. Three sub-iterations are employed after every global iteration, to minimize the linearization and factorization errors. The no-slip, adiabatic-wall condition is applied on the surface of the blade. Periodic conditions were invoked along the azimuthal and span-wise directions to simulate the three-dimensional flow in the linear cascade. All the mean quantities presented here are obtained by time-averaging the solution over 7 characteristic times.

After the baseline solution is obtained, a flow separation control strategy using VGJs is implemented on the LPT stator blade. The mean flow for the baseline case was found to separate at about 69% $C_{ax}$, as can be seen in Fig. 39. Based on this observation, VGJs were placed at in a range of 63.5% to 67% $C_{ax}$. All the jet parameters, i.e., blowing ratio, pitch angle, skew angle and duty cycle ratio are kept same as for the cylinder flow, while the $F^+$ value of 1.33 is employed in the present case. The non-dimensional time-period of the pulsing jet corresponding to $\Delta t = 0.0005$ is 0.428, which is equivalent to
1500 time steps. The jet is issued with duty cycle ratio of 50%, which means that the jets are active for the first 750 time steps, and inactive for subsequent 750 time steps.

5.5.1 Time-Averaged Surface Pressure coefficient (Cp) for LPT blade

Cp is evaluated in the present work using the expression:

\[ C_p = \frac{P - P_{\text{ref}}}{\frac{1}{2} \rho_{\text{ref}} U_{\text{ref}}^2} \]

where \( P \) is the local static pressure, and the reference quantities \( P_{\text{ref}}, \rho_{\text{ref}} \) and \( U_{\text{ref}} \) are the flow conditions at the inlet boundary. The variation of time-averaged coefficient of pressure along the blade surface is shown in Fig. 35 which also includes the experimental results of Huang et al (2003). The trend of the numerical Cp curve agrees very well with the experimental data. But, as a whole, the Cp curve does not match with the experimental data and it is observed that, if the y-axis for the Cp curve obtained through numerical computations is shifted by a constant distance, it matches well with the experimental results. It is explained in a study by Mutnuri (2003), that, with additional iterations the computed Cp curve approaches the experimental Cp curve and that the simulation has to be continued for quite a long time for the two curves to match exactly. Later on, it is shown that if the upstream distance of the grid is moved from to 7 \( C_{ax} \), and the grid density to 6.4 million grid points approximately, the simulation reached the stable state at shorter computation execution time, whereas all other simulations could not reach this stage even after running for longer computational execution time.

Another issue with the Cp curve is the presence of oscillations on the upper half of the Cp curve representing the coefficient of pressure distribution over the suction surface of the blade. This issue is addressed in a study of two-dimensional simulations by Mutnuri
(2003). It is observed that the oscillations on the suction surface of the blade could be reduced with grid refinement. Hence, it is believed that the oscillations on the upper half of the curve corresponding to the suction surface of the blade could be reduced with grid refinement.

5.5.2 Total Velocity Comparison

Figures 37 and Fig. 38 show the comparison of the velocity magnitude profiles with the LES results of Rizzetta and Visbal (2003), at the two specified locations on the blade’s surface, for the baseline LPT case. The pressure profiles compare fairly well with that of Rizzetta and Visbal (2003) for the results closer to the surface of the blade, while they don’t away from the blade’s surface. For the present study, a static inflow boundary condition and an extrapolated outflow boundary condition is used while an extrapolated inflow and fixed outflow boundary condition is used, by which accounts for upstream influence in a subsonic flow. This difference in inflow/outflow boundary condition is believed to be one of the important factors responsible for the difference in the velocity magnitude profiles. Another reason for this discrepancy can be the difference in grid size employed in the two studies. The results by Rizzetta and Visbal (2003) are obtained on a grid containing a total of 18 million grid points, while a total of 2 million grid points are employed in the present numerical study. This significant difference in the grid size and their associated truncation errors are believed to be responsible for the differences in the velocity magnitude profiles.

5.5.3 Vorticity Contours and Surface Cp distribution

Flooded contours of time-averaged spanwise vorticity for baseline as well as controlled flow are shown in Figs. 39-40. The baseline mean flow separates at about
70% $C_{ax}$, while the implemented flow separation control strategy totally eliminates the separation. The $C_p$ distributions for controlled and uncontrolled cases are compared in Fig. 41. It can be clearly seen in the figure that, due to the control, the pressure on the upper surface is modified in such a manner that an earlier pressure recovery occurs, increasing the pressure force in the direction of the motion of the blade. The energy analysis performed for the cylinder case, as mentioned in section 5.4.6, is also carried out for the LPT cascade problem, and shows that 17 units of energy issued into the flow save 100 units of energy.

5.5.4 Streamwise Velocity Fluctuations ($u'^2$)

Flooded contours of streamwise velocity fluctuations, for the controlled and the uncontrolled flow, are shown in Figs. 41 and 42, respectively. As shown in Fig. 41, for most part of the upstream portion of the blade, the flow remains laminar as is expected at low Re of 25,000. Once the flow separates on the suction side of the blade it undergoes transition and for a short region, it will become turbulent. Figure 42 clearly shows that the high fluctuations in the region very near to the trailing edge are eliminated by the application of the flow separation control.

5.5.5 Total pressure

To quantify the wake reduction, the integrated wake pressure loss coefficient ($C_w$) is calculated for the controlled and the uncontrolled cases. The same definition of $C_w$ is used as in section 5.4.5, to quantify the wake reduction. $C_w$ is then integrated in the y-direction across the profile. It is found that $C_w$ for the baseline flow is 0.338, and $C_w$ for controlled flow is 0.246, representing 27.5% reduction in loss coefficient.
Chapter 6

Summary and Conclusions

A higher-order accurate numerical scheme, namely, the fourth-order accurate compact-difference scheme with sixth-order filtering (C4F6), has been used to study the flow separation control using Vortex Generator Jets for the cylinder and cascade flow. For this study, the high-order accurate, parallel, Chimera, LES version of the FDL3DI flow solver has been used. The flow control simulations have been performed for two different geometries: circular cylinder and LPT blade. For all the simulations, a multi-block structured grid approach is used. To reduce the total simulation turn-around time significantly, the available massively parallel system is used. Also, a full 3-D simulation is performed for all the above mentioned cases, using the Implicit Large Eddy-Simulation Technique (ILES), to study the effect of the third spatial dimension on the process of separation and transition. An O-topology is used for the grid for cylinder geometry while a combination of O-H topologies is employed for the grid for the LPT geometry.

As a validation case for the parallel flow solver, the flow over a circular cylinder with a multi-block structured grid at $Re_D = 3,900$ is studied first. The results are compared with the numerical results of Kravechenko (2003). The simulation results for the multi-block structured grid using the C4F6 scheme agree well with the published results of Kravechenko (2003).

The flow over a circular cylinder at $Re = 13,400$ is studied next for the model wake problem of flow past a circular cylinder in a moderate Re regime and a preliminary solution is obtained, employing a relatively coarse grid. Next, the simulation is performed on a finer grid to obtain a baseline solution for the flow separation control
study. This is followed by application of the flow separation control using VGJs on the upper surface of the cylinder at about 75° from the stagnation point. The jets are issued into the flow with a blowing ratio of 2.0 and were pitched and skewed by 30° and 70° respectively. An $F^+$ of 1.0 is used along with 50% duty cycle. The cylinder flow at $Re = 13,400$ displays presence of a wide range of vortical structures in the wake region. The separating shear layers are subject to spanwise instability, which leads to the formation of an unsteady and three-dimensional wake, with the characteristic features of typical turbulent flow. It is observed that after the jets are being turned on, the pressure on the surface of the cylinder redistributes in a way so as to reduce the pressure drag significantly. The total pressure loss coefficient and momentum thickness are calculated in the wake at $x/D = 3.0$ and $x/D = 5.0$ and is found to reduce by 10% and 30% respectively.

The simulation for the flow through a low-pressure turbine (LPT) cascade at $Re_{Cax} = 25,000$ is then conducted to provide a baseline solution. Finally, VGJs are incorporated for the LPT cascade flow. VGJs are placed in a range of 63.5% to 67% $C_{ax}$. All the jet parameters, i.e., blowing ratio, pitch angle, skew angle and duty cycle ratio are kept same as the cylinder case, while the $F^+$ value of 1.33 is employed for the LPT cascade problem. The flow exhibited some oscillatory behavior, especially near the suction surface of the blade. The amplitude of the oscillations seen in the $-C_p$ curve on the suction side were reported to reduce significantly, with increase in the grid density and increase in the distance of the upstream inflow boundary location, in the study by Mutnuri (2003). The flow control simulations for the LPT cascade flow reveals 27%
reduction in total pressure loss coefficient along with the total elimination of separation upon application of VGJs.

The CPU time can become very large for high-order accurate simulations. Using the multi-block structured grid approach the simulation turn-around time has been significantly reduced by using the available massively parallel systems.

6.2 Recommendations for Future Work

During the course of this study, several ideas emerged, which require further detailed investigation. The first recommendation is to conduct a full, turbulent flow analysis (with and without control) with more points in z-direction for the LPT cascade problem. For a turbulent analysis, standard Large-Eddy Simulation (LES) approach should be used and that should provide insight into the complex flow features associated with turbulence. Large-Eddy Simulation models the small scales and resolves the large scales of the flow, and provides a realistic solution on a practical grid size. This approach is better suited when the mode of transition and the region where the flow undergoes transition become the key aspects of the analysis. A high-order accurate LES solution will serve as a benchmark solution for flow through the LPT blade linear cascade. This solution can then be used to assess other models such as the k-ε model, and enhance their capability. The second recommendation is to conduct a parametric study by varying various VGJ parameters (hole size, frequency of pulsation, hole placement location, etc.) to determine how the reduction of separated region is affected by these parameters.
REFERENCES


Figure 1. Snapshot of the 8-block structured grid used for $Re_D = 3,900$. 
Figure 2. Coefficient of pressure on the upper surface of the cylinder (Spanwise- and time-averaged over 50 characteristic time units)
Figure 3. Streamwise velocity on the centerline in the wake region (spanwise- and time-averaged)
Figure 4. Streamwise velocity profiles at three locations in the wake region (spanwise and time-averaged)
Figure 5. Crossflow velocity profiles at three locations in the wake region (spanwise and time-averaged)
Figure 6. Instantaneous $z$-vorticity at mid $z$-plane, after 200 characteristic time units for $Re = 3,900$.

Figure 7(a). Contours of spanwise Reynolds stress for $Re = 3,900$. 
Figure 7(b). Contours of Turbulence kinetic energy for Re = 3,900.

Figure 8(a). Probe locations where time history is tracked for Re = 3,900.
Figure 8(b). Streamwise velocity time history for Re = 3,900

Figure 9. Contours of vorticity magnitude for Re = 13,400, depicting formation of Karman Vortex street.
Figure 10. Contours of spanwise Reynolds stress for Re = 13,400.
Figure 11. Contours of Turbulence kinetic energy for Re = 13,400
Figure 12(a). Streamwise velocity profiles at three locations in the wake region (spanwise and time-averaged)
Figure 12(b). Crossflow velocity profiles at three locations in the wake region (spanwise and time-averaged)
Figure 13(a). Streamwise velocity contours for Re = 13,400 after 200 characteristic times

Figure 13(b). Spanwise velocity contours for Re = 13,400 after 200 characteristic times
Figure 13(c). Contours of positive (upper) and negative (lower) crossflow velocity for Re = 3,900 after 200 characteristic time units.
Figure 14. Coefficient of pressure on the upper surface of the cylinder (spanwise- and time-averaged over 50 characteristic time units)
Figure 15. Streamwise velocity on the centerline in the wake region (spanwise- and time-averaged)

$L_t/D = 1.06$ (Present)

$Re = 13400$

Present Simulation (Coarse grid)
Figure 16. Snapshot of the fine-grid showing the block-layout and the grid used for the cylinder simulation.
Figure 17. Instantaneous z-vorticity contours for Re = 13,400, after 20 characteristic time
Figure 18. Grid overlaid on z-vorticity contours for the coarse grid and fine grid (Re = 13,400).
Figure 19. Instantaneous vorticity magnitude contours for $Re = 13,400$ from 0.5 to 10
Figure 20. Lift coefficient time history for cylinder flow at Re = 13,400.
Figure 21. Q-criterion isosurfaces (Q = 4) for cylinder flow at Re = 13,400.
Figure 22. Affect of grid-refinement on streamwise velocity profiles at three locations in the wake region (spanwise and time-averaged), for Re = 13,400.
Figure 23. Affect of grid-refinement on crossflow velocity profiles at three locations in the wake region (spanwise and time-averaged), for Re = 13,400.
Figure 24. Jet orientation and parameters.
Figure 25. Comparison of coefficient of drag for controlled and uncontrolled case.
Figure 26. Comparison of coefficient of lift for controlled and uncontrolled case.
Figure 27. Time history of pressure drag and viscous drag coefficient for controlled and uncontrolled flow.
Figure 28. Comparison of time- and spanwise-averaged surface coefficient of pressure for controlled and uncontrolled case.
Figure 29. Total pressure profile at x/D = 3, for controlled and uncontrolled case for circular cylinder.
Figure 30. Total pressure profile at x/D = 5, for controlled and uncontrolled case for circular cylinder.
Figure 31. Total pressure profile at $x/D = 7$, for controlled and uncontrolled case for circular cylinder.
Figure 32. $U_{avg} (1 - U_{avg})$ variation with $y$, at $x/D = 5$, for controlled and uncontrolled case for circular cylinder.
Inlet

Outlet

Flow entering at an angle of
55° to the vertical

Figure 33. Flow configuration for flow through LPT linear cascade
Figure 34(a). Multi-block structured block layout of the computational domain with 4-point overlaps.

Figure 34(b). Multi-block structured grid in the computational domain.
Figure 35. Comparison of time- and spanwise-averaged surface coefficient of pressure for present simulation, with experimental results of Huang.
Figure 36. Comparison of time- and spanwise-averaged surface coefficient of pressure for present simulation, with experimental results of Huang after shifting.
Figure 37. Comparison of velocity magnitude profile for present simulation at $x/D = 0.75$, with numerical results of Rizzetta.
Figure 38. Comparison of velocity magnitude profile for present simulation at $x/D = 0.85$, with numerical results of Rizzetta.
Figure 39. Time- and spanwise-averaged z-vorticity contours for $\text{Re} = 25,000$ (uncontrolled case).
Figure 40. Time- and spanwise-averaged z-vorticity contours for Re = 25,000 (controlled case).
Figure 41. Comparison of surface coefficient of pressure for controlled and uncontrolled case.
Figure 42. Contours of streamwise velocity fluctuation (uncontrolled case).

Figure 43. Contours of streamwise velocity fluctuation (controlled case).
Vortices past the cylinder at Re = 13400 (without control)

[click the image to play the clip]
Vortices past the cylinder at $Re = 13400$ (with VGJ control)

[click the image to play the clip]