High-Reynolds Number Flow Past a Rotating Cylinder With and Without Thom Discs

A dissertation submitted to the University of Manchester for the degree of Master of Science by Research in the Faculty of Engineering and Physical Sciences

2011

Nicholas D. Johnson

School of Mechanical, Aerospace and Civil Engineering
## Contents

- **Contents** ........................................................................................................................................... 1
- **List of Figures** ................................................................................................................................. 4
- **List of Tables** ................................................................................................................................. 9
- **Nomenclature** ................................................................................................................................. 9
- **Abstract** ........................................................................................................................................ 12
- **Declaration** .................................................................................................................................. 13
- **Copyright Statement** ..................................................................................................................... 14
- **Acknowledgments** .......................................................................................................................... 15

### 1 Introduction

1.1 Basic Flow Past a Rotating Smooth Cylinder ............................................................................. 19
1.2 Outline of the Dissertation .......................................................................................................... 20

### 2 Literature Review

2.1 Introduction .................................................................................................................................. 22
2.2 Flow Past a Rotating Smooth Cylinder ......................................................................................... 23
  2.2.1 Early Work and Theories ......................................................................................................... 23
  2.2.2 Laminar Flow Regime ............................................................................................................ 24
  2.2.3 Turbulent Flow Regime ........................................................................................................ 29
2.3 CFD Models Used for Rotating Smooth Cylinder Flows ............................................................ 33
  2.3.1 CFD Models Used for Laminar Flow .................................................................................... 33
  2.3.2 CFD Models Used for Turbulent Flow .................................................................................. 36
2.4 Co-Rotating Disc Flow .................................................................................................................. 39
  2.4.1 Overview of flow behaviour .................................................................................................. 39
  2.4.2 CFD models used .................................................................................................................. 41
2.5 The Flettner Rotor ......................................................................................................................... 42
2.6 Summary of previous work .......................................................................................................... 45

### 3 Governing Equations

3.1 Navier-Stokes Equations .............................................................................................................. 49
3.2 Reynolds-Averaged Navier-Stokes Equations ............................................................................ 49
3.3 Turbulence Model ......................................................................................................................... 50
  3.3.1 Reynolds stresses .................................................................................................................... 50
  3.3.2 High-Re $k – \varepsilon$ model .................................................................................................... 51
Contents

3.4 Near-Wall Region ............................................................................................. 53
  3.4.1 Log-law formulation ................................................................................. 54
  3.4.2 Advanced Log-Law Function ................................................................. 55
3.5 Summary .......................................................................................................... 56

4 Numerical Methods ............................................................................................... 58
  4.1 Discretization Scheme .................................................................................. 58
  4.2 Convection Scheme ...................................................................................... 59
    4.2.1 UMIST scheme ....................................................................................... 60
  4.3 Time Discretization ...................................................................................... 60
  4.4 Boundary Conditions .................................................................................... 61
    4.4.1 Fluid Inlet ............................................................................................... 62
    4.4.2 Wall ......................................................................................................... 62
    4.4.3 Fluid Exit ............................................................................................... 62
    4.4.4 Periodic boundary ................................................................................. 62

5 Axial Effects on Rotating Smooth Cylinder Flows ............................................. 64
  5.1 Preliminary Remarks ...................................................................................... 64
  5.2 Grid Resolution .............................................................................................. 65
  5.3 Axial Effects of Flow Structure ..................................................................... 66
    5.3.1 Mean Near-Wake Profile Structure ....................................................... 66
    5.3.2 Low Rotation Span Length Dependency .............................................. 66
    5.3.3 High Rotation Span Length Dependency ............................................. 67
  5.4 Axial Effects of Aerodynamic Performance .................................................. 68
  5.5 Summary of Findings .................................................................................... 70
  5.6 Figures .......................................................................................................... 71

6 Flettner Rotor Flows with Thom Discs ............................................................... 81
  6.1 Preliminary Remarks ...................................................................................... 81
  6.2 Grid resolution ............................................................................................... 82
  6.3 Thom Disc Affected Flow Behaviour .......................................................... 82
    6.3.1 Temporal Changes in Flow Past the Flettner Rotor ................................ 83
    6.3.2 Fluid Motion on Cylinder-Normal Planes ............................................. 84
    6.3.3 Unsteadiness within the Thom Disc Cavity ........................................... 85
  6.4 Thom Disc Affected Aerodynamic Properties .............................................. 86
    6.4.1 Rotor Lift Effects .................................................................................... 87
6.4.2 Rotor Drag Effects ................................................................. 88
6.4.3 Magnus Lift Comparison ......................................................... 88
6.5 Figures ...................................................................................... 90

7 Concluding Remarks and Future Work ........................................... 102
7.1 Effects of Span Length on Solution Domain ................................. 102
7.2 Effect of Thom Discs on the Flettner Rotor ................................. 104
7.3 Future Work .............................................................................. 105

Final Word Count: 24,501
List of Figures

Figure 1.1: Anton Flettner's prototype rotor ship nicknamed Buckau, built 1925-26. ... 16
Figure 1.2: The E-Ship 1 developed during 2008-10 by Enercon and Lindenau GmbH/Cassens Werft. .......................................................... 17
Figure 1.3: Von Karman vortex street behind a stationary cylinder; Mittal and Kumar (2003). ................................................................. 19
Figure 1.4: Top: flow past non-rotating cylinder, Re=1000, α=0; bottom: shows the translating stagnation point, T₁, for rotating disc flow; Re=1000, α=3; dashed lines denote streamwise centreline (Badr et al 1990). ........................... 20
Figure 2.1: Experimental designed and used to capture the near-wake development (from Coutanceau & Ménard (1985)). .......................................................... 23
Figure 2.2: Streamlines displaying the formation of the vortex to be shed as well as the vortex which translates to the front of the cylinder; Re = 200, α = 3 (from Badr et al. (1990)). ........................................................................ 24
Figure 2.3a: Vorticity contours for dimensionless time (e) t=35 to (h) t=54; Re = 200, α = 3.25 (from Chen et al. (1993)). .............................................. 25
Figure 2.4: Lift coefficient as a function of dimensionless time (t) for various α, Re = 200 (from Mittal & Kumar (2003)). ........................................ 26
Figure 2.5: Image of near-wall wake behind the cylinder, Re = 200, a = 2.07 (from Coutanceau & Menard (1985)). .............................................. 27
Figure 2.6: Vorticity field for various α at dimensionless time (t) where the lift coefficient is largest; Re = 200 (from Mittal & Kumar (2003)). ................... 29
Figure 2.7: Comparison of numerical and experimental values of time-averaged Strouhal number against α; various Re (from Aoki & Ito (2001)). .......... 30
Figure 2.8: Mean recirculation region behind non-rotating cylinder in laminar and turbulent flow; Re =3900, 140,000. Note the reattachment angle for turbulent flow is much steeper than for laminar (from Breuer (2000)). .................. 31
Figure 2.9: Lift coefficient against α; Re = 140,000 (from Karabelas (2010)). .......... 32
Figure 2.10: Outline of geometry for O-type mesh regarding laminar flow past a rotating circular cylinder (from Kang et al. (1999)). ....................... 34
Figure 2.11: Mesh geometry and density for turbulent flow past a rotating cylinder. (from Karabelas (2010)). ......................................................... 36
Figure 2.12: Computed and measured axial variation of the radial component of velocity in a co-rotating disc flow, top - r/R=0.65, mid - r/R=0.75, bot - r/R=0.85; Re = 146,000 (from Randriamampianina et al. (2004)).

Figure 2.13: Various axial slices of the turbulent kinetic energy for flow between co-rotating discs; Re = 146,000, aspect ratio = 0.5. s = distance between discs, x = axial position; note the flow is axisymmetric at each position (from Zacharos (2009)).

Figure 2.14: Streamlines of turbulent flow between co-rotating discs with RSM; Re = 146,000. horizontal axis is distance between discs, vertical axis is radial position along discs (from Randriamampianina (2004)).

Figure 2.15: Coefficient of lift, drag, and torque against rotation rate for flow past rotating cylinder with evenly spaced Thom fences (from Thom (1934)).

Figure 2.16: Lift-drag plot for rotors of various aspect ratios (λ) and disc diameter-cylinder diameter ratios (κ). L/D = length/cylinder diameter from (Norwood (1996)).

Figure 3.1: Inner sub-layers of near-wall boundary layer with streamwise velocity profile. Transition point found at point where velocity profile and viscous sub-layer curves meet.

Figure 5.1: Cylinder geometry and orientation where the z-direction (not shown) is the cylinder span, in each case the cylinder spins counter-clockwise.

Figure 5.2: (left) A single x-y plane cross-section of 3D smooth cylinder mesh. (right) Near-wall mesh created using y+ and employed the log-law wall function.

Figure 5.3: (left) 3D C-type smooth cylinder mesh with Z_{max} = 2R, note the cylinder surface is highlighted in purple. (right) 3D C-type smooth cylinder mesh with Z_{max} = 6R, note the cylinder surface is highlighted in purple.

Figure 5.4: Mean turbulence kinetic energy near-wake profiles compared with Karabelas (2010).

Figure 5.5: Mean velocity field streaklines for α = 2, compared with Karabelas (2010) and 2D computations.

Figure 5.6: Mean velocity field streaklines for α = 5 compared with 2D computations. Refer to section 5.3.3 for explanation regarding the appearance of crossed streaklines.

Figure 5.7: 2D diagram of slices made through the cylinder at given y values. Flow is left to right, diagram corresponds to successive figures.
Figure 5.8: Streamwise contours of W/U for $\alpha = 5$ and $Z_{max}/R = 2$ at $t = 100$. Flow is left to right, labels on left corresponds to diagram in figure 5.6. ...................... 75

Figure 5.9: Streamwise contours of W/U for $\alpha = 5$ and $Z_{max}/R = 6$ at $t = 100$. Flow is left to right, labels on left corresponds to slices in figure 5.6. ......................... 76

Figure 5.10: Temporal development of streamwise contours of W/U for $\alpha = 5$ and $Z_{max}/R = 6$, planes corresponds to position A in figure 5.6. Flow comes from left, labels on left correspond to non-dimensional time, $t$. .......................... 77

Figure 5.11: 3D velocity field streaklines with vorticity contouring of the flow over the cylinder (using the orientation in fig. 5.1, $\omega = \alpha$). ........................................ 78

Figure 5.12: Distribution of the pressure coefficient for all smooth cylinder cases in the present study, also includes purely 2D equivalent cases and results from Karabelas (2010) ($\theta$ measured according to figure 5.1). ............................. 78

Figure 5.13: Lift coefficient comparison as a function of spin ratio is graphically displayed. Present model computational results are given as data points due to how few of each case were simulated. Lines connecting these points would not best represent the lift coefficient behaviour with spin ratio. ......................... 79

Figure 5.14: Temporal evolution of the coefficient of lift for the present 3D smooth cylinders cases, the corresponding 2D cases, and a $Re = 800k$ with $\alpha = 5$ from Craft et al. (2011). .......................................................... 79

Figure 5.15: Temporal evolution of the coefficient of drag for the present 3D smooth cylinders cases and the corresponding 2D cases. ........................................ 80

Figure 6.1: Near-wall mesh resolution for (left) the 4R-TD mesh and (right) 6R-TD mesh. .................................................................................................................. 90

Figure 6.2: 3D mesh resolution for rotor with Thom discs for (left) 4R-TD and (right) 6R-TD meshes. ......................................................................................... 90

Figure 6.3: Representation of the geometry and orientation for the simulated cavity found between a single set of Thom discs. .................................................. 91

Figure 6.4: A particular temporal region of unsteadiness in the lift coefficient from $t = 150$ to $t = 175$. Labels indicate positions in time which will correspond to successive figures in this chapter. .................................................. 91

Figure 6.5: Velocity field past the rotor for $D/R = 6$ at the geometric mid-plane between the discs $z/R = 0.5$ for temporal positions defined by figure 6.4. The outer ring indicates the outer edge of the disc. .................................................. 92
List of Figures

Figure 6.6: Velocity field for D/R = 4 on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond for figure 6.4. ................................................................. 93

Figure 6.7: Velocity field for D/R = 6 on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond for figure 6.4. ................................................................. 94

Figure 6.8: Velocity field for D/R = 4 on cylinder normal planes at $\theta = 0$ and $\pi$. Labels correspond for figure 6.4. ................................................................. 95

Figure 6.9: Velocity field for D/R = 6 on cylinder normal planes at $\theta = 0$ and $\pi$. Labels correspond for figure 6.4. ................................................................. 96

Figure 6.10: Plotted turbulence energy for D/R = 4 on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond with figure 6.4; range given is for mean k data. ... 97

Figure 6.11: Plotted turbulence energy for D/R = 6 on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond with figure 6.4; range given is for mean k data. ... 98

Figure 6.12: Plotted turbulence energy for D/R = 4 on cylinder normal planes at $\theta = 0$ and $\pi$. Labels correspond with figure 6.4; range given is for mean k data. ... 99

Figure 6.13: Plotted turbulence energy for D/R = 6 on cylinder normal planes at $\theta = 0$ and $\pi$. Labels correspond with figure 6.4; range given is for mean k data. ..... 100

Figure 6.14: Temporal evolution of the lift (solid lines) and drag (dashed lines) coefficients both present simulations with Thom discs for Re = 140k, also includes results from Craft et al (2011) for Re = 800k; all cases with $\alpha = 5$. ... 100

Figure 6.15: Mean pressure contours at mid-span ($z = 0.5$) for (left) D/R = 4 and (right) D/R = 6............................................................................................................. 101

Figure 6.16: Mean cylinder wall pressure distribution. $\theta$ follows the orientation in figure 6.3, reference pressure chosen for $Cp = 1$ at $\theta = 0$ for $\alpha = 0$. ................. 101

Figure A.7.1: Plotted P for D/R = 4 on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond with figure 6.4................................................................. 110

Figure A.7.2: Plotted P for D/R = 6 on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond with figure 6.4................................................................. 111

Figure A.7.3: Plotted P for D/R = 4 on cylinder normal planes at $\theta = 0$ and $\pi$. Labels correspond with figure 6.4; range given is for mean P data.......................... 112

Figure A.7.4: Plotted P for D/R = 6 on cylinder normal planes at $\theta = 0$ and $\pi$. Labels correspond with figure 6.4; range given is for mean P data.......................... 113

Figure A.7.5: Spanwise motion (W/R) around the rotor for D/R = 4 at z/R = 0.5 (per geometry in figure 6.3) for temporal positions defined by figure 6.4. The outer ring indicates the outer edge of the disc. ................................................................. 114
Figure A.7.6: Spanwise motion (W/R) around the rotor for D/R = 6 at z/R = 0.5 (per geometry in figure 6.3) for temporal positions defined by figure 6.4. The outer ring indicates the outer edge of the disc. ........................................................... 115
List of Tables

Table 2.1: Overview of all combinations of high-Re simulations and computed parameters from Breuer (2000) ................................................................. 37
Table 2.2: Summary of aerodynamic results and key observations from previous studies. ................................................................................................................. 48
Table 5.1: Resolution of both grids used in the smooth cylinder simulations. .......... 65
Table 5.2: Coefficients of lift and drag. Compared with purely 2D flow, and the results from Karabelas (2010) and Craft et al. (2011) (note: Craft results are for Re = 800,000). ............................................................................................................. 69
Table 6.1: Resolution of both grids used in the simulations of rotating cylinder flows with Thom discs. ................................................................. 82
Table 6.2: Mean lift and drag coefficients for present study and Craft et al. (2011)..... 88

Nomenclature

\[ \alpha \] rotor rotation rate (cylinder wall tangential velocity/free-flow velocity)
\[ \alpha_{\text{cr}} \] critical rotor rotation rate
\[ \delta_{ij} \] Kronecker delta
\[ \Delta \] denotes a change in
\[ \varepsilon \] energy dissipation
\[ \Gamma \] fluid diffusion coefficient
\[ \kappa \] Von Karman constant
\[ \kappa' \] modified Von Karman constant
\[ \kappa_d \] diffusion coefficient
\[ \lambda \] aspect ratio
\[ \sigma \] turbulent Prandtl number
\[ \sigma_{\varepsilon} \] constant
\[ \sigma_{\kappa} \] constant
\[ \Omega \] cell volume
\[ \rho \] fluid density
Nomenclature

\[ \theta \] azimuthal angle
\[ \phi \] transport variable
\[ \tau \] shear stress
\[ \tau_w \] wall shear stress
\[ \mu \] dynamic viscosity
\[ \mu_t \] effective viscosity
\[ \nu \] kinematic viscosity
\[ \nu_t \] turbulent viscosity
\[ \gamma \] mass diffusion coefficient
\[ A \] cell face area
\[ c_{e1}, c_{e2} \] constants
\[ c_\mu \] proportionality constant
\[ c_d \] coefficient of drag, aka Cd
\[ c_l \] coefficient of lift, aka Cl
\[ c_p \] coefficient of pressure
\[ d \] characteristic length
\[ D \] disc diameter
\[ e, w, s, n \] cell face descriptors
\[ E \] empirical constant
\[ E^* \] empirical constant
\[ f \] some function
\[ i,j,k \] nodal discretization indicators
\[ k \] turbulence kinetic energy
\[ k_f \] kinematic energy of the random fluctuations
\[ n \] temporal discretization indicator
\[ \alpha \] see \[ \alpha \]
\[ P, E, W, S, N \] cell centre indicators
\[ P \] pressure
\[ \bar{P} \] mean pressure
\[ P_k \] energy production rate
\[ Pe \] Peclet number \([((d * U)/\gamma)]\)
\[ R \] cylinder radius
\[ Re \] Reynolds number \([2UR/\nu])\)
Nomenclature

$S_{cv}$  collective source terms
$S^P_{cv}$  collective source at centre node
$St$  Strouhal number
$t$  dimensionless time (time*(U/R))
$u,v,w$  fluctuating components of velocity
$u^2,v^2,w^2$  normal stresses
$U,V,W$  instantaneous components of velocity
$\bar{U},\bar{V},\bar{W}$  mean components of velocity
$U^+$  non-dimensional velocity, $U/(\tau_w/\rho)^{1/2}$
$U^*$  non-dimensional velocity, $Uk^{1/2}/(\tau_w/\rho)$
$u_i u_j$  Reynolds stress
$x, y, z$  cartesian coordinates
$y^+$  non-dimensional distance to the cylinder wall, $y(\tau_w/\rho)^{(1/2)}/\nu$
$y^*$  non-dimensional distance to the cylinder wall, $yk^{1/2}/\nu$
$Z_{disc}$  length between successive Thom discs
$Z_{max}$  length of the axial cylinder span
2D  two-dimensional
3D  three-dimensional
CDS  central difference scheme
CFD  computational fluid dynamics
EVM  eddy viscosity model
LES  large eddy simulation
N-S  Navier-Stokes
PANS  partially-averaged Navier-Stokes
QUICK  quadratic upstream weighted interpolation for convection kinematic
RANS  Reynolds-averaged Navier-Stokes
RBL  rotating boundary layer
RSTM  Reynolds stress transport model
STREAM  semi-implicit method for pressure-linked equations
UDS  upwind difference scheme
UMIST  upstream monotonic interpolation for scalar transport
URANS  unsteady Reynolds-averaged Navier-Stokes
Abstract

The present dissertation was written based on a computational fluid dynamics study of high-Reynolds number flow past a few different geometries of the Flettner rotor with and without Thom discs. The three-dimensional unsteady Reynolds-averaged Navier-Stokes (URANS) equations were solved and discretized with a finite volume approach. Two separate types of flows were investigated; a 3D smooth cylinder flow investigating two different cylinder span lengths and a rotor with fixed circumferential discs (Thom discs) to investigate the idea introduced by Thom (1934).

A high-Re \( k-\varepsilon \) eddy viscosity turbulence model resolved the turbulence while the near-wall motion was solved using an advanced log-law wall function. The simulations are run as if the rotor was instantaneously translated and rotated simultaneously.

The 3D smooth cylinder simulations studied the span length dependencies of the cylinder’s flow behaviour and aerodynamics. Two solution spaces were generated differing by span length. The grids modelled flows for \( \text{Re} = 140,000 \) and dimensionless rotation rates \( \alpha = 2 \) and 5. The study revealed the three-dimensionality in the flow behaviour past the cylinder at both rotation rates and the formation of periodic spanwise undulation along the cylinder when the rotation rate is increased. Likewise, lift and drag coefficients were investigated where the smooth cylinder was found to be aerodynamically independent of span length.

Thom discs of infinitiesmally small thickness were fixed on the cylinder to investigate the possibility of aerodynamic improvements, as proposed by Thom but not convincingly detected in a preceding CFD study. The solution space models a single region between two discs referred to as the Thom disc cavity. The two simulations model flow for \( \text{Re} = 140,000 \) with \( \alpha = 5 \), while of lower \( \text{Re} \) than those a Flettner rotor would typically experience, it was believed to be sufficiently high, nonetheless. A qualitative analysis of the flow behaviour revealed the fluid motion within the Thom disc cavity was highly complex and highly random in nature. This study found that a great deal of aerodynamic instability was exhibited as the radius of the Thom discs was increased.
Declaration

No portion of the work referred to in the dissertation has been submitted in support of an application for another degree or qualification of this or any other university or other institute of learning.
Copyright Statement

The author of this dissertation (including any appendices and/or schedules to this dissertation) owns any copyright in it (the “Copyright”) and he has given The University of Manchester the right to use such Copyright for any administrative, promotional, educational and/or teaching purposes.

Copies of this dissertation, either in full or in extracts, may be made only in accordance with the regulations of the John Rylands University Library of Manchester. Details of these regulations may be obtained from the Librarian. This page must form part of any such copies made.

The ownership of any patents, designs, trade marks and any and all other intellectual property rights except for the Copyright (the “Intellectual Property Rights”) and any reproductions of copyright works, for example graphs and tables (“Reproductions”), which may be described in this dissertation, may not be owned by the author and may be owned by third parties. Such Intellectual Property Rights and Reproductions cannot and must not be made available for use without the prior written permission of the owner(s) of the relevant Intellectual Property Rights and/or Reproductions.

Further information on the conditions under which disclosure, publication and exploitation of this dissertation, the Copyright and any Intellectual Property Rights and/or Reproductions described in it may take place is available from the Head of School of Mechanical, Aerospace and Civil Engineering.
I would like to extend my sincerest gratitude to my supervisors Prof. B.E. Launder, Dr. T.J. Craft and Prof. H. Iacovides for their tremendous support and guidance throughout this research and dissertation process. They were always there when needed help with the many preliminary CFD projects and the eventual CFD research carried out, and offering comments and corrections in my dissertation write-up. I am honoured to have known and worked with such distinguished and intelligent individuals, and I wish them all the best in the future.

I would also like to thank Mr. S. Hood for his invaluable help with the various computing issues that I encountered throughout my research. He has provided me a tremendously generous service.

Last and certainly not least I would like to thank my family for giving me the strength and drive to pursue all my goals in life. Without their love and support through the highs and lows of this course, I could not have completed this program.
Chapter 2: Introduction

1 Introduction

The drive to make Earth a so-called “greener” planet has brought up many new concepts and technologies adopted around the world. One of the main areas of concern has always been alternative fuels in the wake of the diminishing oil supply and the harmful chemicals being put into the atmosphere from burning such a substance. In the 1920’s German Engineer, Anton Flettner, theorized using the Magnus effect of a rotating cylinder to be used as a means for propulsion. A Flettner rotor ship was designed and built in 1925-26, figure 1.1, with the intention to provide an improvement to the conventional sailing rig (Norwood 1996). The rotor proved successful after a number of sea trials with the Buckau as it could sail in much harsher conditions than a traditional sailing rig due to its greatly reduced surface area thus reduced drag. However, with the rotor’s lack of ability to sail with headwinds or tailwinds and the emergence of cheap fossil fuels and the modern diesel engine, Flettner’s efforts were pushed aside. As of late, interest in these rotors has resurfaced as the desire for environmentally friendly alternatives to the world’s oil dependence becomes an increasing priority.

The well-known Magnus effect phenomenon can be observed in a flow past a rotating cylinder by way of a measured perpendicular lift force on the cylinder itself. It was Sir Isaac Newton who was the first to adequately describe the Magnus effect while watching tennis in Cambridge (Gleick 2004). Later in the 19th century, Magnus became the first to officially theorize the phenomenon and thus it was named after him. Thom (1934) found an increase in the lift force when evenly-spaced circumferential discs are placed along the length of the rotor. It is this case in which the present work is most concerned with and plans to expand on Thom’s findings. Plenty of previous work has
been carried out in the field of translating and rotating smooth cylinder flows, and has provided an adequate knowledge base for other applications.

The Flettner rotor as mentioned has resurfaced in the minds of many as a sailing alternative. In 2006, a number of students from the University of Flensburg with the guidance of Professor Lutz Feisser built a Flettner rotor driven proa. The vessel was unveiled and christened at the annual maritime festival Flensburg Nautics. The vessel was named the Uni-Kat Flensburg and used as a demonstration piece to show the abilities of a vessel fitted with a Flettner rotor. The small proa housed a solar-powered electric motor to rotate the single cylinder atop the vessel. The team concluded the efficiency of the Flettner rotor was an improvement by a factor of ten over the conventional sailing rig (Möller, 2006).

In 2008, Enercon, a wind energy company, designed and began construction on a Flettner powered cargo ship, named the E-Ship 1, figure 1.2, to transport wind turbines among other products the company manufactures. This ship has 4 individual 25 metre tall Flettner rotors, two fore and two aft, to utilize the wind energy. The original construction was done by Lindenau Gmbh shipyards but in the early part of 2009, the ship was transferred to Cassens Werft to finish the job (E-Ship 1, 2009). The ship was completed in April 2010 and successful sea trials were completed by the end of July 2010 (Kennedy (2010)). According to Kennedy (2010), the E-Ship 1 completed its maiden cargo carrying voyage to Dublin, Ireland on 10 August, 2010. The rotors are used to assist the diesel engine, also aboard the E-Ship 1, and cut fuel consumption by approximately 30%.

A much bigger proposal was made by Salter (2007) which called for the construction of around 1500 purpose built radio-controlled Flettner rotor vessels. The
vessels are to be used to spray sea water into the air to increase the solar reflectivity in the clouds over the oceans. This process is known as cloud seeding or cloud albedo control. Flettner rotors are desirable for this project as they are easier to control remotely than conventional sailing rigs and the rotors require a reported five to ten percent of the power required by a conventional engine. Salter (2007) proposed the power supply be regenerated through turbines being dragged through the water similar to the Soloman Technologies electric-boat system. An initial investment from The Discovery Channel was offered to build a smaller prototype version of the proposed vessel in Salter (2007).

As it will be clear in the literature review, the physics of a translating and rotating cylinder flow has been studied for a number of years both by experimental processes and numerical simulations. The literature review outlines results from laminar and turbulent smooth cylinder flow, but no numerical studies have been carried out regarding a cylinder with evenly-spaced Thom discs thus the flow behaviour is relatively unknown until the present work. The main focus of this work is the flow behaviour of the present case as well as the three-dimensionality of a turbulent rotating cylinder flow. A finite volume solution will be used to discretize the Reynolds Averaged Navier-Stokes equations and the turbulence will be solved using a high Re k-ε model.

The objectives of this study are:

- To study 3D rotating cylinder flows with different cylinder span lengths and discuss the resulting flow structure.
- To compare the aerodynamic properties of each case and discuss the effects and dependencies of the cylinder span length.
- To carry out combinations of Thom disc spacing and disc-cylinder diameter ratio and compare with previous studies to obtain a reasonable understanding of the flow structure and their effects of lift and drag.
1.1 Basic Flow Past a Rotating Smooth Cylinder

The basic flow past a rotating cylinder has been well documented through experimental and numerical studies. It is known that a non-rotating cylinder produces a Von Karman vortex street behind the cylinder shown in figure 1.3. The character of the non-rotating flow is governed by the Reynolds number, Re, and for all Re within the laminar flow regime the flow is essentially two-dimensional.

![Von Karman vortex street behind a stationary cylinder](image)

*Figure 1.3: Von Karman vortex street behind a stationary cylinder; Mittal and Kumar (2003).*

Flow past a rotating cylinder shows a fairly different behaviour than that observed in non-rotating case. The rotation adds another parameter $\alpha$, and it is defined as the ratio of the cylinder tangential velocity to the uniform flow velocity. As the cylinder rotates within a uniform cross flow, one side of the cylinder perpendicular to the uniform flow will move against the flow and the opposite with move with the flow. A difference in velocity from one side of the cylinder to the other will essentially mean a pressure drop across the cylinder is present. The pressure difference, much like one present over an airfoil, forces the flow to bend toward the low pressure side (side where cylinder wall is moving with the uniform flow) and destroying the time-averaged streamwise symmetry found about the centreline in the non-rotating case. The dashed lines in both diagrams of figure 1.4 denote the centreline through the cylinder parallel to the upstream flow. The asymmetry found means that the stagnation point at the leading edge of the cylinder, $T_1$ in figure 1.4, rotates in the direction of the rotation as well as moving away from the cylinder surface with increasing $\alpha$. It has been found as you will see further in the literature review that a rotating cylinder may still produce a Von Karman vortex street but only up to a critical rotation rate, $\alpha_c$, where the vortex shedding disappears. The critical rotation rate is dependent on Re and the fluid properties.
Chapter 1: Introduction

With the asymmetry comes a pressure drop across the perpendicular axis to the uniform flow. The side of the cylinder where the cylinder wall tangentially moves opposite the free flow creates an adverse pressure gradient and thus an area of higher pressure than the pressure in the far field. The opposite side of the cylinder sees an opposite effect where an area of low pressure forms since the cylinder wall moves with the fluid and reduces drag across that section of the cylinder. This phenomenon creates a similar effect to that found on an airfoil where a lift force is acted on the high pressure side of the cylinder in the direction of the low pressure side. This phenomenon is known as, and noted earlier, the Magnus effect.

1.2 Outline of the Dissertation

The present dissertation will examine the topic of three-dimensional high-Re flow past the Flettner rotor by first showing why the topic is important, then by discussing the methods used to obtain the data being presented, and then by presenting the results and drawing necessary conclusions. Chapter 2 provides a literature review which compiles relevant previous knowledge on the present topic and related material that may aid in
Chapter 1: Introduction

the comprehension of the present study material. Once a good knowledge base has been established, chapter 3 outlines and defines the governing equations that will model the flow and its turbulence motion. In chapter 3 the Reynolds-averaged Navier-Stokes equations will be introduced and discussed as well as an appropriate turbulence model to model the Reynolds stresses that arise from averaging the basic Navier-Stokes equations. The issue of near-wall modelling is also addressed in Chapter 3. Chapter 4 discusses the methods used to take the governing equations and apply them to the meshes used in the present study. Chapter 4 also discusses and justify the mesh used in the present study. Chapter 5 will be the first of the results chapters which looks at rotating smooth cylinder flows and the effects that adjusting the cylinder span length has on the flow behaviour and aerodynamic performance. Chapter 6 examines flow past the Flettner rotor with the addition Thom discs. The effects the addition of axial discs has on the flow structure and the rotor’s aerodynamic performance is discussed. Chapter 7 is a summary of the major findings which will also include concluding statements and suggestions for future works.
2 Literature Review

2.1 Introduction

The Flettner rotor was a concept theorized by a German engineer by the name of Anton Flettner. He proposed placing rotatable cylinders with axes vertical on the deck of a ship and rotating them. As a cylinder is rotated a perpendicular lift force is created and this is known as the Magnus effect. The ensuing future work will look at a variation of the Flettner rotor which studies the flow effects of placing evenly-spaced circumferential discs along the axis of the cylinder.

This chapter discusses many closely related subtopics in relation to the Flettner rotor as there are not very many published works entirely devoted to the Flettner rotor itself. Flow past a circular cylinder has been researched for many years in regards to the widely known Von-Karman vortex street found for a flow past a non-rotating cylinder. Rotating cylinder flows have also been researched extensively over the last 100 years starting with work carried out by Prandtl (1925). In the last 20 years or so, many have used the increased abilities of computers to model this type of flow instead of performing costly experiments. Work has been published for laminar and turbulent flow past a rotating cylinder. For the purpose of this work the turbulent case is of most relevance. This chapter will not only look at the results from these various works but also the numerical methods used.

This chapter begins both by looking into the various experimental and numerical cases of rotating cylinder flows in the laminar and turbulent regimes. A great deal of interest is in these works given the sheer amount of data and information obtained from them. This part is split into two sections, laminar flow and turbulent flow, with each section broken into key engineering topics. The emphasis of the first part’s entirety is on the studies of turbulent flow by numerical methods. The second part of the paper will compare the numerical models used for smooth rotating circular cylinder flows. The third part briefly looks into co-rotating disc flows. This topic is of particular interest as the present research studies flow over circular cylinders with evenly spaced circumferential discs, also known as Thom discs. Lastly there is a relatively brief portion devoted to the Flettner rotor due to the small amount of literature available. The
main focus in this last part is the results obtained for a cylinder with evenly spaced discs along its axis.

2.2 Flow Past a Rotating Smooth Cylinder

2.2.1 Early Work and Theories

The earliest relevant work done was by Prandtl (1925) when he carried out an experiment of flow over a circular cylinder and the Magnus effect found as the cylinder rotates while uniformly translating through a flow media. He argued the maximum lift coefficient generated, due to the Magnus effect, had a limit of $4\pi$ or roughly 12.6 (Prandtl 1925). Prandtl & Tietjens (1934) investigated rotating cylinder flow to discover the pressure distribution around the cylinder, but the instrumentation available, including the apparatus used, did not allow for the complex flow field to be captured. The experimental investigation of Coutanceau & Ménard (1985) was the first study which allowed the flow field to be observed and adequately captured with the apparatus in figure 2.1. They looked at the near-wake area behind a rotating cylinder for $Re \leq 1000$, but concluded the near-wake region did not change for $200 \leq Re \leq 1000$. Coutanceau & Ménard found a way to take pictures of the flow by placing suspended particles in the fluid, figure 2.5. They discovered the rotation of the cylinder induced a continuous fluid layer on the cylinder wall, now known as the rotating boundary layer (RBL), which increased with $\alpha$ but decreased with increasing $Re$. They confirmed the flow symmetry was destroyed by the rotation and was related to the fluid-to-wall velocity distributions. Further investigation on flow symmetry and the rotating boundary layer was found for $Re = 140,000$ by Karabelas (2010) whose results will be discussed later in this review.
2.2.2 Laminar Flow Regime

Flow behaviour and stability
Badr et al. (1990) recreated the work Coutanceau & Ménard did five years earlier and compared it to numerical results they carried out. This work provided a comparison of experimental measurements to theoretical results from numerical solutions of the Navier-Stokes equations. The experimental apparatus described by Badr et al. (1990), figure 2.1, is similar to that of Coutanceau & Ménard (1985). They studied the flow for $10^3 \leq \text{Re} \leq 10^4$ and $\alpha \leq 3$, which saw the two sets of results agree with each other but only up to the limitations of the experimental apparatus. For the case of $\text{Re} = 1000$ and $\alpha = 3$, two vortices formed; one vortex is shed and the second slips to the front of the cylinder and is quickly crushed by the stream as can be seen in figure 2.2. The numerical solution adopted in Badr et al. approaches a stable steady-state solution, but their experimental results showed the flow becomes turbulent. Chew et al. (1995) ran the same case using a hybrid vortex scheme, explained later in this chapter, which confirmed the numerical results from Badr et al. (1990). Coutanceau & Ménard (1985) found the rate of detachment of the eddies (vortices shed) accelerated with increasing $\alpha$. This statement remained true up to a critical rotation rate $\alpha_L$. For $\alpha \geq \alpha_L$ they argued only one eddy was present and the second essentially disappeared and $\alpha_L \sim 2$ for all Re. The critical rotation rate was given an exact value of $\alpha_L = 2$ by Chew et al. (1995) for $\text{Re} = 1000$, ...
and vortex shedding is completely suppressed at $\alpha \geq 3$. A stable solution was found in the numerical results of Badr et al. (1990) for $\alpha = 3$, which supported the findings by Coutanceau & Ménard (1985). A very low Re flow investigation was published from Kang et al. (1999), who saw stable flow for $0 \leq \alpha \leq 2.5$ when $\text{Re} < 40$, but can be unstable for $\text{Re} > 60$. Ingham (1983) had published the first results for $\text{Re} < 40$ and never saw any vortex shedding for all $\alpha \leq 0.5$, thus proving the findings of Kang et al. (1999) are consistent. They also found $\alpha_L$ to be in good agreement with previous works and the periodic oscillations did disappear at $\alpha_L$. On a contrary note, Chen et al. (1993) argued that the $\alpha_L$ was not found at $\alpha = 2$. It was discovered that vortex shedding was still present with $\text{Re} = 200$ at $\alpha = 2$ and 3.25. These findings have been refuted by many successive studies and they were finally put to rest by Mittal & Kumar (2003) when

they not only disproved Chen et al. (1993), but narrowed in on the value of $\alpha_L$ to be 1.91 for $\text{Re} = 200$. Mittal & Kumar (2003) was able to debunk the argument of Chen et al. (1993), figure 2.4, by concluding that they simply just didn’t run the solution for enough time to allow the flow to fully develop into a steady state behaviour.

Mittal & Kumar (2003) found good agreement in flow characteristics and eddy behaviour with previous work. They ran their solutions for dimensionless time $t = 225$ for $\alpha \leq 2.07$ and up to $t = 300$ for all $\alpha \geq 2.07$. They discovered the flow stabilizes for $1.91 \leq \alpha \leq 3.45$, but a secondary region of instability was found from $4.34 \leq \alpha \leq 4.75$. The instability in this region was stronger but only included one vortex that is shed clockwise (negative) from the upper side instead of positive vortices (counter-clockwise...
Chapter 2: Literature Review

from upper side) like the vortices shed from both sides for \( \alpha \geq 1.9 \). Mittal & Kumar (2003) concluded positive vorticity is fed close to the stagnation point where the fluid is slow moving. This strong positive vorticity is then turned into recirculating fluid and the positive vortex builds up until it is diffused to the outer flow field and shed. Craft et al. (2010) confirmed the stability results of Mittal & Kumar while proving out their numerical methods. The study reran many of Mittal & Kumar’s simulations and they too found the phenomenal second region of instability. However, El Akoury et al. (2009) found a much larger range of \( \alpha \) where the secondary region of instability provided large pulsation. For \( \text{Re} = 300 \) they found instability from \( 3.9 \leq \alpha \leq 4.8 \) suggesting it had a Re dependency.

Along with the argument over \( \alpha_L \), a second area where previous studies have disagreed with each other is with the Strouhal (St) number, a dimensionless quantity used to describe the oscillating flow. Kang et al. (1999) found it to increase with increasing Re but decrease with increasing \( \alpha \). This is contrary to Chew et al. (1995) and Diaz et al. (1983) who claimed St increased with \( \alpha \).

The stability of flow over a rotating circular cylinder is usually tracked by looking at the coefficient of lift, seen here in figure 2.4. As previously noted Prandtl (1925) concluded that the maximum lift coefficient that could be reached was \( 4\pi \) (~12.6). Chew et al. discovered the magnitude of the lift coefficient confirmed Prandtl’s limit of \( 4\pi \) for \( 0 \leq \alpha \leq 6 \). The experimental investigation of Tokumaru & Dimotakis (1993) at \( \text{Re} = 3.8 \times 10^3 \) and the numerical solution from Mittal & Kumar (2003) for \( \text{Re} = 200 \) found this limit to be untrue for \( \alpha \geq 3.25 \).
Chapter 2: Literature Review

Flow symmetry and other areas of interest
The asymmetry of a time-averaged uniform flow over a rotating cylinder was one of the first behaviours noticed that differed from flow over a non-rotating cylinder. As a cylinder is in rotation the flow symmetry is essentially destroyed by the pressure drop across the two halves of the cylinder, the upper and lower halves in this case. This was captured by Coutanceau & Ménard (1985) shown in figure 2.3. They determined that the upset in flow symmetry was due to the differing fluid-to-wall velocities on the upper and lower sides of the cylinder. Badr et al. (1990) modelled the same flow and gave a visual display of the symmetry and what happens to the stagnation point as α increased. The stagnation point was found to move around the circumference of the cylinder and away from the cylinder wall. Figure 2.2 shows how the stagnation point (labelled T₁) translates clockwise from the centreline around the cylinder. The RBL increases with increasing α which pushes the stagnation even further toward the far field. In certain cases the stagnation point has been found to be as far as 90 degrees clockwise around the cylinder which can be seen in Mittal & Kumar (2003). In this investigation, they claim the stagnation point can be as far as roughly two cylinder diameters from the cylinder wall. The changes in flow symmetry as α increases can be seen in figure 2.6; one can better understand the severe shift in stagnation point location seen in Mittal & Kumar (2003) for the higher rotation rates. This behaviour was confirmed in Padrino & Joseph (2004).

For the sake of topic completeness, Chew et al. found the pressure coefficient distribution on the cylinder surface decreases with increasing α, and the RBL increases with increasing α. In a similar study, Padrino & Joseph (2004) used various methods to
find the thickness of the vortical region around the cylinder. The work found agreeable efforts within the study, but there is no corresponding works which confirm their findings. Mittal & Kumar (2003) looked into the power coefficient and found it increases rapidly with increasing $\alpha$. They concluded that the power required to generate the lift force on a circular cylinder by use of Magnus effect was too expensive to be an adequate substitute for any propulsion application. Lastly it was reported by Ingham (1983) that the lift coefficient increased linearly with increasing $\alpha$ for $\alpha \leq 0.5$; it was also found that the drag coefficient decreased with increasing $\alpha$. These conclusions are supported up to $\alpha \leq 2$ for most succeeding works (Badr et al. (1990), Chew et al. (1995), Kang et al. (1999), Mittal & Kumar (2003)). In a 3D laminar study from El Akoury et al. (2009) investigated spanwise motions along a cylinder where the solution space modelled a cylinder with a span length that was much longer than the cylinder diameter. The study found a non-rotating cylinder generates a spanwise undulation which diminished with increased $\alpha$ up to $\alpha = 1.5$. The present study determines the undulation effects in turbulent flows with higher rotation rates.
Figure 2.6: Vorticity field for various $\alpha$ at dimensionless time ($t$) where the lift coefficient is largest; $Re = 200$ (from Mittal & Kumar (2003)).

2.2.3 Turbulent Flow Regime

Early concerns with turbulence flow
Studieos of high Re flow over rotating circular cylinders have not been too extensively investigated in the past with more of the industrial applications being in the laminar flow regime. The aim of these investigations was to acquire a better understanding of the mechanisms of turbulent flows thus created. According to Cantwell and Coles (1983), a non-rotating cylinder flow study, large eddy was an organized concentration of large-scale vorticity which is energized through entrainment while able to retain its geometry. These large eddies they refer to are now known as vortices that are shed from the near-wake behind a cylinder in uniform flow. It was Cantwell and Coles objective to study the shedding of these vortices through the then available experimental
methods. Though they were not successful in acquiring more knowledge of turbulence mechanics, they did however discover that there was some degree of three-dimensionality to flow over a cylinder once it becomes turbulent. The investigation did not incorporate any rotation to the circular cylinder; its usefulness for future development was in discovering the three-dimensional characteristics of high-Re flow.

Three-dimensionality requirement
The case of flow past a circular cylinder is well documented in laminar 2D scenarios. The transition into turbulent flow brings greater fluctuation and greater interaction between large-scale and small-scale eddies. Cantwell and Coles (1983) discovered and concluded that as the flow transitions into turbulent flow it becomes three-dimensional and this was confirmed numerically by Breuer (1998, 2000). He used large eddy simulation (LES) to model flow over a non-rotating cylinder at Re = 3900 and was able to display the inaccuracy of the 2D simulations compared to the same solution in 3D. Aoki & Ito (2001) also found disagreement in a joint numerical and experimental investigation. The lift and drag characteristics did not agree well between the three-dimensional experiment and the two-dimensional CFD solution.

Flow stability
Diaz et al. (1983) carried out an experimental investigation for flow at Re = 9000 and $\alpha \leq 2.5$. They used a wind tunnel and a variable-speed motor to simulate a rotating and translating cylinder. They made the length of the cylinder much larger than the diameter so the properties were essentially functions of the streamwise and cross-stream components. With a set-up like the one briefly described the time-development of the flow around the cylinder could not be tracked and therefore the results obtained are for large time $t$. They were able to conclude that the flow does stabilize for $\alpha \geq 2$ by plotting the turbulent fluctuations in the streamwise and cross-stream directions. For much higher $Re$, Aoki & Ito (2001) plotted the time-dependent development of the lift coefficient for a flow of $Re = 50,000$ and found a lack of stability for all flows where $\alpha$
\( \leq 1 \), but the solution was only run for \( t = 20 \) which from previous laminar flow studies is assuredly not enough time to derive a strong conclusion on flow stability. Good agreement was found in Karabelas (2010) who reran the same case as Aoki & Ito (2001). Karabelas (2010) modelled flows for \( \alpha \leq 2 \) and found the flow begins to stabilize with \( \alpha \geq 1.3 \) for \( \text{Re} = 140,000 \). Craft et al. (2010) reran many of the cases from the Mittal & Kumar (2003) study and captured a secondary region of instability at \( \alpha = 4 \) which was both weaker in intensity and less frequent than the laminar results found.

As the flow transitions to the turbulent regime, other factors play a part in the flow behaviour such as random turbulent fluctuations. Karabelas (2010) found a way to determine the total resolved kinetic energy of the fluctuations (\( k_f \)). Two components make up \( k_f \), the periodic vortex shedding oscillations and the random turbulent fluctuations. Although the two were not separated to determine the precise contribution of each, it was concluded that \( k_f \) will follow one of three cases:

1. For low \( \alpha \): \( k_f \) will be dominated by the periodic oscillations
2. For moderate \( \alpha \): \( k_f \) will share relatively equal dominance between both
3. For high \( \alpha \): \( k_f \) will be dominated by the turbulent fluctuations

The previous laminar flow studies looked into the behaviour of the Strouhal number and that seems to be appropriate for the turbulent cases as well. Diaz et al. (1983) found \( \text{St} \) for each rotation rate and determined that \( \text{St} \) increased with \( \alpha \). Later numerical studies showed good agreement and confirmed \( \text{St} \) increased with increasing \( \alpha \) for \( \alpha \leq 1 \) (Aoki & Ito (2001), Elmiligui et al. (2004)). Figure 2.7 shows this relationship between \( \text{St} \) and \( \alpha \) from Aoki & Ito (2001). The Strouhal number describes the vortex street formed on the backside of the cylinder wall, but as \( \alpha \) increases the shedding dissipates. Karabelas
(2010) found the vortex initially formed in the viscous layer for low $\alpha$ (~0.5) and did so due to the no-slip condition employed, but the RBL has been found to increase with $\alpha$. The relationship of the RBL, $\alpha$, and the vortex shedding behaviour would be an interesting case to look into for future work. The reattachment angle for $\alpha = 0$ was of concern in works by Karabelas (2010) and Breuer (2000), but this was more a parameter of comparison for the numerical method employed by each. It is interesting to note the recirculation length in turbulent flow is much shorter (steeper reattachment angle) in turbulent flow than laminar flow as can be seen in figure 2.8 from Breuer (2000).

**Lift and drag characteristics**

Tokumaru & Dimotakis (1993) completed an experimental work where they were able to come up with a method to estimate the average lift acting on a rotating circular cylinder in a uniform flow with the use of a water tunnel. Their analysis was based on an inviscid point-vortex model and the measured transverse velocity for $\text{Re} = 3.8 \times 10^3$ for rotation rates in the range $0 \leq \alpha \leq 6$. The work claimed that the lift coefficient limit argued by Prandtl (1925) could be exceeded for $\alpha > 5$. This was also the first experimental investigation which confirmed that increases in the cylinder’s aspect ratio would increase the lift on the cylinder.

Aoki & Ito (2001) carried out a similar experimental investigation for flow at $\text{Re} = 60,000$ and were able to determine the coefficient of lift for $0 \leq \alpha \leq 1$; numerical analysis by Elmiligui *et al.* (2004) shows good agreement with the experimental results. Aoki & Ito (2001) numerically looked at the same case as well but found their
numerical results substantially exceeded the experimental results. Karabelas (2010) used an LES model of the same flow and found good agreement from the numerical results of Aoki & Ito (2001). The disagreement between the results obtained by Elmiligui et al. (2004) and those found by Aoki & Ito (2001) and Karabelas (2010) cannot be explained, but this study will aim to acquire its own results to compare with the previous works (section 5.4). Under the same investigation Karabelas (2010) found the lift coefficient as a function of \( \alpha \) was lower than those found in laminar flow studies for all \( \alpha \leq 2 \) (figure 2.9). Good agreement came from Craft et al. (2010) where they found the generated lift was lower in turbulent flows, but still exceeded Prandtl’s limit which was promising for their study of the Flettner rotor. A second study from Craft et al. (2011) analysed the modelling abilities of two separate turbulence models and the results agreed well with Karabelas (2010) depending on which model was used. A TCL model (discussed later in this chapter) found consistently higher lift through \( \alpha \leq 1 \), but slightly less lift at \( \alpha = 2 \). A second model found good agreement through much of the range, but distinctly less lift for \( \alpha = 2 \). The drag was lower with both models compared to Karabelas (2010).

### 2.3 CFD Models Used for Rotating Smooth Cylinder Flows

#### 2.3.1 CFD Models Used for Laminar Flow

Early models used

Computational Fluid Dynamics is a progressive field of study that is aimed at obtaining results in fluid flow to cut costs otherwise accrued during experimental studies. An early work by Ingham (1983) used a finite differencing scheme to model very low Re (stable) flow over a rotating cylinder. The study used upwind, downwind, and central differencing schemes (CDS) to model the flow. It found the best results with the CDS model since it is of higher-order accuracy. He solved the stream function equation in steady state using cylindrical coordinates to make for an easier solution.

The finite differencing scheme was later implemented again in Badr and Dennis (1985) to follow the time-dependent flow past a rotating and translating cylinder. The
flow was described in terms of the stream function and the scalar vorticity equations. This work used CDS for the spatial discretization and the Crank-Nicolson scheme for the time discretization. This work was able to capture accurate flow behaviour and movement of the stagnation point. The same scheme was used again for different flow conditions in Badr et al. (1990). A similar approach was used in Kang et al. (1993) but solved the two-dimensional primitive variable Navier-Stokes (N-S) equations. They used a fully-implicit, fractional step method in time (Crank-Nicolson) with CDS in space. They solved the nonlinear discretized equations with a Newton method. The mesh used was aptly termed an O-type mesh (see figure 2.10) which they felt was best for modelling the case of oscillatory rotation of a cylinder since it was a small mesh which meant it was cheaper to use, but still accurately captured the downwind flow behaviour past the cylinder.

![Figure 2.10: Outline of geometry for O-type mesh regarding laminar flow past a rotating circular cylinder (from Kang et al. (1999)).](image)

**Advancements in CFD models**

Around the same time Chen et al. (1993) used a different method to solve the same two-dimensional primitive variable N-S equations in an attempt to better model the wake behind a rotating cylinder. They implemented a new formulation of the Biot-Savart law to integrate a velocity/vorticity formulation of the N-S equations and get around the complications at the boundaries. Finite differencing was used in space where CDS was utilized in the radial direction and a pseudospectral transform was used in the circumferential direction. To advance the flow in time, a fully explicit method was employed with the use of the Runge-Kutta method to advance the vorticity field.
temporally. This approach generated good data, but was not run for a sufficient amount of time to reach asymptotic conditions and thus the paper had little impact.

Chew et al. (1995) combined two schemes over two different regions of the solution space to model a generic case at Re = 1000 and higher rotation rates. First they designed their mesh with two regions in mind. The first region is the near-wall viscous region around the cylinder and the second region is all of the far-field area. A hybrid-vortex scheme was designed which combined the use of the diffusion-vortex method in the near-wall region and the vortex-in-cell method in the far field. The diffusion-vortex method was used since it produces good lift and drag data as well as reduces CPU time. The method solves the linear diffusion equation and the inviscid convection equation separately with the two different methods. This derived method is of a finite volume type method. It was found that the resulting data had a slight dependence on the mesh density around the cylinder in the near-wall region because it was difficult to estimate where the near-wall region ends. Therefore, the diffusion-vortex method is grid dependent and in most cases dependent on the flow conditions. No time stepping scheme was stated in the article.

Recent models used for laminar flow
The most recent and thorough study published is that of Mittal and Kumar (2003) which made attempts to clear up any disagreements among previous works. In this study, a stabilizing finite element method was used to solve the primitive variable N-S equations. Within a rectangular solution space, multiple meshes were tested and the mesh which gave best agreement with previous data was selected for continued use. The model carries out a global linear stability analysis by using the streamline-upwind/Petrov-Galerkin (SUPG) and the pressure-stabilizing/Petrov-Galerkin (PSPG) methods. The nonlinear equations resulting from the finite element discretization were solved by using the generalized minimal residual (GMRES) technique. Lastly the time stepping, when employed, was accomplished using the implicit method which allows the unsteady terms simply to be resolved when desired. Using this model, Mittal and Kumar (2003) settled some disagreements among previous studies as well as discover new areas of instability within certain ranges of $\alpha$. El Akoury et al. (2009) used a numerical method based on a pressure-velocity formulation with a predictor-corrector pressure scheme. Temporal discretization was an alternating direct implicit formulation adopted from a Peaceman and Rachford scheme.
Chapter 2: Literature Review

The emergence of very competent commercial CFD packages has been increasing in popularity for research work. Padrino & Joseph (2004) used FLUENT with the QUICK and PISO schemes to model flow-Re flows to quantify the thickness of the vortical region in various Re and $\alpha$.

### 2.3.2 CFD Models Used for Turbulent Flow

**Large eddy simulation models**

The various early studies of turbulent flow over a cylinder were done with a non-rotating cylinder. This was until Breuer started looking at his case and wrote various articles on the subject. Breuer (1998) outlines the CFD model and turbulence model used in each of the published articles. Breuer (1998, 2000) used an LES model to simulate the three-dimensional, time-dependent, N-S equations. During LES the equations are filtered to separate the large scale and small scale motions. The LES code used was based on a 3D finite volume method for non-orthogonal body fitted grids, and it used CDS for the spatial discretization. The temporal discretization used by Breuer (1998, 2000) was a predictor-corrector scheme which employed a Runge-Kutta method for integrating the momentum equations. An incomplete LU decomposition method was employed for the pressure corrections, and the overall time stepping was done with the explicit method which is stated to work well for LES with small time steps to resolve the turbulence motion. In LES a subgrid scale model (a characteristic length scale) is necessary, as Breuer (1998) stated, to solve the subgrid scale stresses which describe the small-scale structures on the larger eddies. Breuer (1998, 2000) utilizes both the Smagorinsky length scale with Van Driest damping and the dynamic model. Several combinations of discretization schemes,
Chapter 2: Literature Review

meshes, and subgrid scale models were tried for best results and can be found in Table 2.1.

A much more recent work used LES and the Smagorinsky length scale to model flow of slightly different flow conditions. Karabelas (2010) employed the finite volume code from FLUENT 6.3 to solve the 3D incompressible time-dependent viscous N-S equations with \( \text{Re} = 140,000 \). The spatial and temporal discretization schemes used were CDS and the fully implicit method respectively. Within each time step, the pressure corrections were solved by an accelerated implicit Gauss-Seidel iteration. Karabelas (2010) obtained results that were in good agreement with previous experimental works by applying the very fine mesh found in Figure 2.11 despite using a commercial code.

Table 2.1: Overview of all combinations of high-Re simulations and computed parameters from Breuer (2000)

<table>
<thead>
<tr>
<th>Run</th>
<th>Grid</th>
<th>( Z_{\text{max}} )</th>
<th>SGS model</th>
<th>( L/D )</th>
<th>( C_s )</th>
<th>( C_{\text{tan}} )</th>
<th>St</th>
<th>( \theta_{\text{app}} (\degree) )</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Coarse grid different SGS models</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>A1</td>
<td>( 165 \times 165 \times 64 )</td>
<td>2D</td>
<td>Dynamic</td>
<td>0.572</td>
<td>1.239</td>
<td>-0.398</td>
<td>0.204</td>
<td>96.37</td>
</tr>
<tr>
<td>A2</td>
<td>( 165 \times 165 \times 64 )</td>
<td>2D</td>
<td>Smag. ( C_3 = 0.1 )</td>
<td>0.416</td>
<td>1.218</td>
<td>-0.411</td>
<td>0.217</td>
<td>95.16</td>
</tr>
<tr>
<td>A3</td>
<td>( 165 \times 165 \times 64 )</td>
<td>2D</td>
<td>Smag. ( C_3 = 0.005 )</td>
<td>0.712</td>
<td>0.707</td>
<td>-0.677</td>
<td>0.247</td>
<td>94.58</td>
</tr>
<tr>
<td>A4</td>
<td>( 165 \times 165 \times 64 )</td>
<td>2D</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>divergent</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Fine grid different SGS models</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>B1</td>
<td>( 325 \times 325 \times 64 )</td>
<td>2D</td>
<td>Dynamic</td>
<td>0.336</td>
<td>1.454</td>
<td>-0.764</td>
<td>0.204</td>
<td>95.00</td>
</tr>
<tr>
<td>B2</td>
<td>( 325 \times 325 \times 64 )</td>
<td>2D</td>
<td>Smag. ( C_3 = 0.1 )</td>
<td>0.375</td>
<td>1.286</td>
<td>-0.480</td>
<td>0.203</td>
<td>92.59</td>
</tr>
<tr>
<td>B4</td>
<td>( 325 \times 325 \times 64 )</td>
<td>1D</td>
<td>–</td>
<td>0.654</td>
<td>0.388</td>
<td>-4.457</td>
<td>0.236</td>
<td>99.33</td>
</tr>
<tr>
<td><strong>Coarse grid different domain sizes</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>C1</td>
<td>( 165 \times 165 \times 64 )</td>
<td>1D</td>
<td>Smag. ( C_3 = 0.1 )</td>
<td>0.509</td>
<td>0.971</td>
<td>-0.083</td>
<td>0.235</td>
<td>96.60</td>
</tr>
<tr>
<td>C2 = A2</td>
<td>( 165 \times 165 \times 64 )</td>
<td>2D</td>
<td>Smag. ( C_3 = 0.1 )</td>
<td>0.416</td>
<td>1.218</td>
<td>-0.411</td>
<td>0.217</td>
<td>95.16</td>
</tr>
<tr>
<td>C3</td>
<td>( 165 \times 165 \times 64 )</td>
<td>2D</td>
<td>Smag. ( C_3 = 0.1 )</td>
<td>0.462</td>
<td>1.276</td>
<td>-0.514</td>
<td>0.218</td>
<td>93.86</td>
</tr>
<tr>
<td><strong>Fine grid different domain sizes</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>D1</td>
<td>( 325 \times 325 \times 64 )</td>
<td>2D</td>
<td>Smag. ( C_3 = 0.1 )</td>
<td>0.413</td>
<td>1.057</td>
<td>-0.221</td>
<td>0.196</td>
<td>93.62</td>
</tr>
<tr>
<td>D2 = B2</td>
<td>( 325 \times 325 \times 64 )</td>
<td>2D</td>
<td>Smag. ( C_3 = 0.1 )</td>
<td>0.375</td>
<td>1.286</td>
<td>-0.480</td>
<td>0.203</td>
<td>92.59</td>
</tr>
<tr>
<td>D3</td>
<td>( 325 \times 325 \times 64 )</td>
<td>2D</td>
<td>Smag. ( C_3 = 0.1 )</td>
<td>0.419</td>
<td>1.368</td>
<td>-0.600</td>
<td>0.205</td>
<td>91.45</td>
</tr>
<tr>
<td><strong>Experiments</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cantwell and Coles (1983)</td>
<td>( \approx 0.44 )</td>
<td>1.237</td>
<td>-0.21</td>
<td>0.179</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Wieselberger et al. (1972), Achenbach (1968), Son and Hanratty (1999), Zdralevich (1997), Fey et al. (1998)</td>
<td>( \approx 1.2 )</td>
<td>( \approx 0.2 )</td>
<td>(see text)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Reynolds averaged Navier-Stokes (RANS) equation models

Other approaches to model turbulent flow over a rotating cylinder involved solving the unsteady Reynolds-averaged Navier-Stokes equations. Aoki & Ito (2001) used a RANS method to model flow for a \( \text{Re} = 60,000 \) and 140,000. The RNG \( k-\varepsilon \) model was introduced as it has been successfully applied for separating flows and transition flows. Like Karabelas (2010), this work used the commercial FLUENT software package and the discrete finite volume method to solve the formulated equations. They used the SIMPLE pressure-correction method to relate the velocity field to the pressure field.
Lastly the Eulerian implicit method discretized the unsteady equations in time. They used a mesh made up of triangular elements which vary in density across the solution space. The numerical results reported did not agree well with the experimental results carried out adjacent in the same study, but do agree fairly well with similar numerical investigations.

The eddy viscosity models (EVM) are sometimes said to be unsuitable for turbulent curved flows, but Elmiligui et al. (2004) offer variations of the RANS approach to give better results with EVMs. The article argues that the RANS approach over predicts the eddy viscosity which results in excessive damping of the unsteady motion present. The investigation uses NASA’s PAB3D code, a structured, multi-block code which uses finite volume to solve the formulated equations and uses a fully-implicit temporal discretization with CDS for the spatial discretization of the formulated equations. Elmiligui et al. (2004) compared the RANS approach to both a hybrid RANS/LES approach and a partially averaged N-S (PANS) equations approach. The hybrid RANS/LES model is used in conjunction with Menter’s SST two-equation model. The approach transitions from the RANS method to LES, and this transition is a function of grid spacing and the turbulent length scale. The PANS method was developed to overcome grid dependency associated with the hybrid RANS/LES method. It solves for the unresolved kinetic energy and the energy dissipation. This method also has a turbulent length scale like the hybrid method as well as the LES method. Elmiligui et al. concluded both the hybrid RANS/LES method and the PANS method showed better agreement with experimental data than the RANS method. The PANS method has not been extensively used but it does show promise for future cases.

Recent studies of turbulent flow past rotating cylinders by Craft et al. (2010, 2011) solves the unsteady RANS (URANS) equations in a pair of finite volume investigations. The studies use a code defined by Lien & Leschziner (2nd-Mom Turb-Trans, 1994) called STREAM with a UMIST limiter also developed by Lien & Leschziner (UMIST, 1994). The study used two different turbulence models, one being a high-Re \( k-\varepsilon \) model and the other was a stress-transport model which shapes the model to comply with the two-component limit (TCL) where turbulence reduces at a wall or free surface. Of the two, the TCL model agreed better with comparative results although the \( k-\varepsilon \) model was promising and cheaper. The second study used essentially the same method except they decided to use a standard log-law wall function as it was determined the near-wall regions within the cavity are very complex and were
not a focus of the study. The TCL model was used and found better definition in the flow compared to the LES data from Karabelas (2010).

### 2.4 Co-Rotating Disc Flow

Flow between two co-rotating, coaxial discs has been well documented from past works, but only within a finite space as usually defined by an outer shroud surrounding the discs. The general flow behaviour and a few past numerical methods are discussed. A look into Ekman layer formation and behaviour between co-rotating discs is desired. Lastly, a brief discussion of the numerical models used for turbulent co-rotating disc flows is discussed. Only past investigations of turbulent flow between co-rotating discs will be discussed as the present study regards flows in the turbulent regime.

#### 2.4.1 Overview of flow behaviour

**Ekman layer formation**

The Ekman layer is a boundary layer which Owen and Rogers (1989, 1995) defines as driven by the centrifugal force the rotating discs has on the fluid in the near-wall region of the disc. For a disc with no centre axial cylinder combining the discs, the Ekman layer is found to start forming at some point radially away from the disc centre and grows along the radius outwards to the edge of the disc where the flow then becomes much more complex.
(Randriamampianina *et al*). The behaviour in the outer radius of the Ekman layer has only been observed for co-rotating discs inside a cavity and experiences upward flow behaviour (or downward if the one is observing the behaviour of the top disc) as it nears the outer shroud. This behaviour is not necessarily expected in the present study due to the uniform flow past the discs.

**Flow behaviour**
The fluid behaviour between the two co-rotating discs is complex and three-dimensional. The profile of the radial velocity can be seen in figure 2.12 and found to be quite similar to that of a wall jet across a flat plate. Randriamampianina *et al.* (2004) and Zacharos (2009) found good agreement with such a behaviour with flows in the turbulent regime. Zacharos (2009) has found that this wall jet profile grows radially and reaches a maximum at about 0.75R-0.85R where R is the radius of the discs. The fluid forced out centrifugally will force an entrainment of fluid back into the centre cavity via the mid-plane between the discs (Randriamampianina *et al.* (2004), Zacharos (2009)).

![Figure 2.13: Various axial slices of the turbulent kinetic energy for flow between co-rotating discs; Re = 146,000, aspect ratio = 0.5, s = distance between discs, x = axial position; note the flow is axisymmetric at each position (from Zacharos (2009)).](image)
Studies have found that the flow is symmetric about the mid-axial plane as found in figure 2.14. The interaction causes large-scale oscillations between the discs that varies radially and axially across the discs, but remains axisymmetric about the entirety of the cavity, figure 2.13. The frequency of these oscillations seems to be dependent upon the axial, azimuthal, and radial position as well as the flow conditions, such as Re and the axial distance between the discs (Herrero (1999), Zacharos (2009)). Small scale turbulent motions were also observed by Zacharos (2009). The present study may not see the overall behaviour described since the uniform cross-stream will be present and is expected to dominate the flow behaviour.

2.4.2 CFD models used

Two numerical studies regarding the case of turbulent flow between co-rotating discs have been discussed. Randriamampianina et al. (2004) looked at flow between co-rotating discs with $Re = 1.46 \times 10^5$ and compared results from the axisymmetric numerical simulation (ANS) and the Reynolds Stress Transport Model (RSTM). He found that the RSTM model agreed better with experimental data obtained in the same work as well as previous studies. The ANS model appeared promising in theory, but did not produce the results that the RSTM model was capable of. Figure 2.14 shows the streamlines of the co-rotating disc flow modelled with the RSTM. Zacharos (2009) used an unsteady RANS approach with a high-Re $k-\varepsilon$ turbulence model to compute the same flow to that found in Randriamampianina et al. (2004). The author was also able to draw close agreement with experimental data obtained in previous works. Zacharos (2009) found large-scale eddy behaviour as well as some small-scale motions in the flow. Randriamampianina et al. (2004) and Zacharos (2009) obtained good agreement with each other, but the studies concluded that further investigation was necessary to acquire more details of the complex flow.
2.5 The Flettner Rotor

The Flettner rotor was pioneered by Anton Flettner in the 1920’s as a revolutionary means of ship propulsion. Today that still seems to be the sole practical application for a concept as such. The Flettner rotor is simply a rotating cylinder placed on its end on the deck of a ship or boat. It is the Magnus force which provides the propulsion for the ship. The original design as noted in Thom (1934) was a smooth cylinder with one disc placed on the cylinder’s ends (of course with one end on the ship deck where airflow will be minimal, the only disc used was placed on the top end). A concept vessel was built in 1924, named “Buckau”, and its initial test runs were successful but it was eventually deemed less efficient than internal combustion engines. In the 1930’s, Thom (1934) looked into placing evenly spaced discs along the length of the cylinder to hopefully increase lift and decrease drag.

Contrary to an earlier statement in this review, Thom (1934) claims Prandtl (1925) found a lift coefficient limit of $2\pi$. Thom (1934) found that placing the fences along the length cylinder did not give much increase to the lift of the rotor, but it did delay the eventual rise in the drag coefficient as $\alpha$ increased. The lift coefficient for such a structure reached values of about 18 (well above Prandtl’s limit) for very high $\alpha$. He also found agreement in previous works of the drag coefficient dropping until $\alpha \approx 7$ where it suddenly steeply increases. Figure 2.15 shows the lift, drag, and torque...
coefficients measured in Thom (1934). He also found better results for lift and drag when the diameter of the discs was three times the diameter of the cylinder as compared to a smaller ratio. Norwood (1996) states the theory behind why the fences provide a reduction in drag is because they prevent much of the axial flow present on a smooth cylinder thus reducing drag along the length of the cylinder. Thom (1934) later went on to measure the torque (figure 2.15) on the cylinder and calculate the power required to rotate the cylinder with discs. The original application was for a small aeroplane with a wing (rotor) one foot in diameter and 12 feet long being rotated 6 times the applied 100 ft/s wind speed. He found the required power of 4830 HP to be too high, thus the idea of rotating a cylinder of actual scale would be impracticable.

Norwood (1996) discussed the use of rotors for wind-driven vessels. He claimed that Thom (1934) reported lift coefficients of over 30 (figure 2.16), but Thom (1934) does not necessarily support this claim. Norwood also provides useful information for Flettner's purpose built vessel “Buckau” which shows a promising outlook for the original Flettner rotor design. Norwood (1996) was able to find a flaw in Thom (1934) regarding the power required to rotate the cylinder. Norwood claims Thom (1934) neglected the scaling laws and thus the calculations for the required power were orders of magnitude too high. Norwood concluded that the power required to drive the rotor was not excessive and discusses a few ideas of how to power the rotation of the cylinder. Further investigation of Norwood (1996) regarding the conclusions made may bring further validity to this area of research.
Chapter 2: Literature Review

The recent work of Craft et al. (2010) numerically investigated turbulent flow past a Flettner rotor fixed with Thom discs for Re = 800,000. The study modelled a single geometry of the Thom disc cavity with a disc diameter of 4R and an axial adjacent disc spacing of 4R as well. Note the quantity R denotes the cylinder radius. Several rotation rates were used investigated up to \( \alpha = 8 \). The study resulted in little future promise in its attempt to legitimize the use of Thom discs. The addition of discs on the bare cylinder appeared to decrease the Magnus lift compared to a smooth cylinder, and increased the drag. Lastly the study concluded the disc were responsible for major instabilities at certain rotation rates where a smooth cylinder showed no periodicity. The study concluded that it unfortunately raised more questions than it answered which is why the authors pushed forward and delivered a second study fairly soon after.

The second study released by Craft et al. (2011) used the same geometry as the previous study but employed a standard log-law wall function to solve near-wall viscous sub-layer knowing that was more convenient yet unreliable in capturing significant features of the near-wall flow. The resulting aerodynamic performance data indicated the addition of Thom disc smooth out the majority of periodic instability for \( \alpha \leq 5 \). For higher rotations the Thom discs returned the same random periodic instabilities found with the equivalent smooth cylinder flow. Marginally higher lift was achieved compared to various past numerical and experimental works, and the drag was comparable although not with the LES drag results from Karabelas (2010). The study put a glimmer of interest back into the topic where several rotor geometries are yet to be investigated.

A comment will be made regarding the torque applied to the rotor and the differences that may be seen from using different disc geometries. Though it is not this study’s intent to investigate the mechanical nature of the rotor, it is noted that it is assumed the required torque on the Flettner rotor correlates approximately with the amount of surface area exposed to the uniform fluid moving past it. Simply put, larger discs or larger numbers of discs in principle should require larger torque. This study also recognizes the uniform velocity (wind speed) may influence the torque requirement as well. In a related issue, the aerodynamics of the rotor will also be affected by the addition of the discs and the present study aims to investigate this further.
2.6 Summary of previous work

Advancements in computational fluids have made complex geometries such as the flows presented in this chapter, easier to model, understand, and thus become more applicable to modern industries. Advancements in understanding the behaviour of laminar and turbulent flow past a rotating cylinder have been compiled and presented. It is known that in almost all cases the flow steadies once the rotational parameter \( \alpha \) reaches a critical rate \( \alpha_L \approx 2 \). Karabelas (2010) found \( \alpha_L \) for flow well in the turbulent regime drops to roughly \( \alpha_L \approx 1.3 \). The flow behaviour for rates less than \( \alpha_L \) are unstable and develops a Von-Karman vortex street in the near-wake behind the cylinder. The vortex oscillations are greatest when \( \alpha = 0 \) and diminish as \( \alpha \) increases until the shedding is completely suppressed at \( \alpha_L \). A few components of the flow are still left unresolved such as the behaviour of the Strouhal number which is still undetermined as to if it increases or decreases with increasing \( \alpha \).

The more important phenomenon of concern to the future work is the appearance of the Magnus effect when a uniform flow is acted upon a rotating cylinder. This effect takes the form of a lift force perpendicular to the cylinder as a function of Re and \( \alpha \). The coefficient of lift and drag were popular quantities measured in previous works and are the intended components to be measured in future work for comparison. Prandtl (1925) concluded the lift on a cylinder reaches but does not exceed a certain limit in any flow. That limit corresponds to a lift coefficient not exceeding \( 4\pi \), though Thom (1934) stated Prandtl’s limit is \( 2\pi \). Prandtl’s work in 1925 could not be acquired by this author so Prandtl’s actual limit is unknown to his knowledge. The limit may be a moot point as some investigations have already shown the limit can be exceeded (Tokumaru & Dimotakis (1993), Chen et al. (1993), and Mittal & Kumar (2003)). Lastly we know that the lift coefficient increases with \( \alpha \) and the drag coefficient initially declines but eventually rises again with increasing \( \alpha \). The exact point where the drag coefficient begins to rise again is uncertain but one opinion can be seen in figure 2.15. The shift in the drag coefficient direction is dependent on the flow conditions.

There have been many numerical studies carried out from finite difference of both the stream function and the N-S equations, to various hybrid models, to RANS, PANS and LES. Most of the models showed good agreement with experimental data as well as each other, but a small issue with the hybrid models as they were somewhat grid
dependent. Most of the numerical works were dependent on a length scale of some type, but for more complex flows like the one in the present work, a dependency on a length scale may not be the best option. Further work could be done to show how newer models such as nonlinear eddy viscosity or stress transport models compare to established data.

A short look into co-rotating disc flows was investigated to understand the typical flow behaviour expected and found that the fluid is forced out near the disc wall due to centrifugal force from the rotation of the discs. The radial velocity profile thus looks much like a wall jet profile. Fluid is then entrained back through the middle of the space between the two discs as mass must be conserved. It is known the flow is symmetric about the mid-axial position (figure 2.14), but does oscillate circumferentially and radially (figure 2.13). Only a select few of numerical works were investigated as they dealt with turbulent flow between co-rotating discs. The present work will also be dealing with flows well into the turbulent regime. The turbulence models used were axisymmetric numerical solution method (ANS), Reynolds stress transport model (RSM), and a RANS approach with a high-Re \( k-\varepsilon \) model. Both the RSM and the PANS methods proved to be adequate and provided good agreement with experimental data. All co-rotating disc flows are done in a cavity and the present work will be a sort of free flow disc flow with a high-Re uniform flow translating through it so will be interesting to see how these two complex flows interact. It is this author’s belief a form of stress transport model will be appropriate for modelling the ensuing very complex flow.

Lastly, a look into the Flettner rotor and past works on this theory, which is the key topic of this work. Not a lot of work has been carried out, nor have any substantially results been reported. It is known that the original idea came about as an alternative method of ship propulsion and a prototype was built in 1924. The work by Flettner himself concluded the rotor was not efficient enough to replace the internal combustion engine. Later work of Thom (1934) looked into placing evenly spaced fences along the length of the Flettner rotor to increase lift and decrease drag. Thom (1934) concluded the discs did not increase the lift by much, but did prolong the drop in drag coefficient with increasing \( \alpha \). However, Thom eventually was claimed the power required to rotate the cylinders was too great for any practical application. This claim was later proven wrong by Norwood (1996) by stating Thom (1934) ignored the scaling laws. The power required was found to be orders of magnitude less thus reintroducing
Chapter 2: Literature Review

the Flettner rotor with Thom’s evenly spaced discs as a feasible means of ship propulsion. Further investigation into disc spacing and diameter is to be carried out, also multiple rotor interacting flows is to be investigated for any unfavourable interactions that may diminish the efficiency and performance. The present work looks to provide more detail in one or more of these areas.
<table>
<thead>
<tr>
<th>Author(s)</th>
<th>Year</th>
<th>Study type</th>
<th>Re</th>
<th>$\alpha$</th>
<th>Aerodynamic Results</th>
<th>Observations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Prandtl</td>
<td>1925</td>
<td>Experimental</td>
<td></td>
<td></td>
<td>Theorized $C_L$ cannot exceed 4$\pi$</td>
<td></td>
</tr>
<tr>
<td>Thom</td>
<td>1934</td>
<td>Experimental</td>
<td>4,500-47,300</td>
<td>0-8.6</td>
<td>adding disc saw lift increase and drag decrease initial and then rise; $C_L$ up to 18.6 for lowest Re, lift decreases with increasing Re</td>
<td>typical disc diameter/cylinder diameter ratio = 3</td>
</tr>
<tr>
<td>Ingham</td>
<td>1983</td>
<td>Numerical</td>
<td>&lt;40</td>
<td>0-0.5</td>
<td>drag decreased with increasing $\alpha$ while lift increases</td>
<td></td>
</tr>
<tr>
<td>Badr et al.</td>
<td>1985</td>
<td>Numerical</td>
<td>200, 500</td>
<td>0.5, 1.0</td>
<td>Found $C_L$ decreased while $C_D$ increased with increasing $\alpha$</td>
<td>Max run time $= 4$ which was later proved insufficient to fully develop the flow</td>
</tr>
<tr>
<td>Coutanceau &amp; Ménard</td>
<td>1985</td>
<td>Experimental</td>
<td>1000</td>
<td>0-3.25</td>
<td>N/A</td>
<td>Short time evolution; no shedding for $\alpha$=2; concluded $\alpha$=2 and not dependent on Re</td>
</tr>
<tr>
<td>Badr et al.</td>
<td>1990</td>
<td>Numerical</td>
<td>1000, 10000</td>
<td>0.5, 1, 3</td>
<td>$\text{Re}=1000$: drag and lift increased with $\alpha$, flow becomes steady for $\alpha$=3; $\text{Re}=10000$: No aero analysis</td>
<td>Simulation run for $t$$\leq$20</td>
</tr>
<tr>
<td>Chen et al.</td>
<td>1993</td>
<td>Numerical</td>
<td>200</td>
<td>0.5-3.25</td>
<td>$C_L$$\approx$12.5 at $\alpha$=3.25, $C_L$ becomes steady at $\alpha$=2.07</td>
<td></td>
</tr>
<tr>
<td>Tokumura &amp; Dimotakis</td>
<td>1993</td>
<td>Experimental</td>
<td>3,800, 6,800</td>
<td>0.5-10</td>
<td>$C_L$$\leq$16; higher aspect ratio yielded higher lift</td>
<td></td>
</tr>
<tr>
<td>Chew et al.</td>
<td>1995</td>
<td>Numerical</td>
<td>1000</td>
<td>0-6</td>
<td>$C_L$$\approx$9 for $\alpha$=4; saw drag increase with $\alpha$, optimum $L/D=2$</td>
<td>$\alpha$=2; periodic oscillations cease at $\alpha$=2</td>
</tr>
<tr>
<td>Kang &amp; Choi</td>
<td>1999</td>
<td>Numerical</td>
<td>60, 100, 160</td>
<td>0-2.5</td>
<td>$C_L$$\leq$8 for $\alpha$$\leq$2.5 at low Re, drag decreases with $\alpha$</td>
<td>$\alpha$ dependent on Re for lower Re; $\alpha$$\approx$2 for $\text{Re}=160$</td>
</tr>
<tr>
<td>Aoki &amp; Ito</td>
<td>2001</td>
<td>Exp &amp; Num</td>
<td>6.0E4, 1.4E5</td>
<td>0-1.2</td>
<td>$C_D$ decreases with increasing $\alpha$; $C_L$ increases with increasing $\alpha$ $0 \leq \alpha \leq 2.5$</td>
<td>Simulation results predicted higher $C_L$ than experimental results</td>
</tr>
<tr>
<td>Mittal &amp; Kumar</td>
<td>2003</td>
<td>Numerical</td>
<td>200</td>
<td>0-5</td>
<td>$C_L$$\approx$27 for $\alpha$$\leq$5, drag agreed well with previous numerical and experimental results</td>
<td>Concluded Chen et al. did not for sufficient time; $\alpha_L=1.91$ for Re=200; secondary unsteadiness observed for $4.35$$\leq$$\alpha$$\leq$4.75; found power needed to rotate cylinder is fairly large</td>
</tr>
<tr>
<td>Elmiligui et al.</td>
<td>2004</td>
<td>Numerical</td>
<td>50,000</td>
<td>0-1</td>
<td>Good agreement with Aoki &amp; Ito; Tokumaru et al.</td>
<td></td>
</tr>
<tr>
<td>Padrino &amp; Joseph</td>
<td>2006</td>
<td>Numerical</td>
<td>200, 400, 1000</td>
<td>3-6</td>
<td>comparable aerodynamic results to Mittal et al.; saw lift and drag change primarily with $\alpha$, not Re</td>
<td>$\text{Re}=500$: shedding suppression at $\alpha$=3; secondary suppression at $\alpha=4.4$ consistent with Mittal, discs smooth out temporal lift unsteadiness</td>
</tr>
<tr>
<td>Craft et al.</td>
<td>2011</td>
<td>Numerical</td>
<td>500, 1.4E5, 8E5</td>
<td>0-8</td>
<td>$C_L$$\approx$25 for $\alpha$$\leq$5 at $\text{Re}=500$; good agreement with Karabelas for higher Re; $\text{CL}=15$ for $\alpha=8$; higher lift and decreased drag with Thom discs for $\alpha$$\geq$5</td>
<td>$\text{Re}=500$: shedding suppression at $\alpha$=3; secondary suppression at $\alpha=4.4$ consistent with Mittal, discs smooth out temporal lift unsteadiness</td>
</tr>
<tr>
<td>Karabelas</td>
<td>2010</td>
<td>Numerical</td>
<td>140,000</td>
<td>0-2</td>
<td>Good lift and drag agreement with Aoki et al., $\alpha_L=1.3$ for $\text{Re}=200$</td>
<td></td>
</tr>
</tbody>
</table>
3 Governing Equations

3.1 Navier-Stokes Equations

The flow in this study is described by the Navier-Stokes (N-S) equations. The continuity and momentum equations conserve mass and momentum for the incompressible flow in the present study. Since the flow is incompressible, the energy and state equations are not necessary to adequately and accurately describe the flow. The equations that govern the flow in the present study are,

\[
\frac{\partial}{\partial x_j}(\rho U_j) = 0 \tag{3.1}
\]

\[
\frac{\partial}{\partial t}(U_j) + \frac{\partial}{\partial x_j}(\rho U_j U_i) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial U_i}{\partial x_j}\right) \tag{3.2}
\]

with \(U_j\) indicating the instantaneous velocity component, \(P\) indicating the instantaneous pressure, \(\mu\) denotes the dynamic viscosity. In the engineering community, these equations will define most flows with the mass and momentum transport described. The three-dimensional turbulent flow past a rotating cylinder is complex which increases in complexity with the addition of the Thom discs. Thus a direct numerical solution of the instantaneous N-S equations would require a great deal of computation time and expense. The present study makes use of the averaged equations which provide a reasonable approximation of the flow.

To simplify the flow calculations, it is quite commonly acceptable to use the time-averaged properties being the instantaneous equations are incredibly difficult to solve. Splitting the instantaneous velocity into a mean component, \(\bar{U}\), and a fluctuating component, \(u\). The resultant equations are called the Reynolds-averaged Navier-Stokes equations (RANS).

3.2 Reynolds-Averaged Navier-Stokes Equations

The Reynolds-averaged equations only incorporate the mean and fluctuating components of velocity. This study, as with many previous investigations, takes advantage of these simplified equations because they are capable of approximating the
solution to produce results comparable with experimental investigations of rotating cylinder flows. The Reynolds-averaged continuity equation in Cartesian notation is expressed by

$$\frac{\partial}{\partial x_i} \left( \rho U_i \right) = 0$$

(3.3)

where $U_i$ is the mean velocity vector with coordinates of $(\bar{U}, \bar{V}, \bar{W})$ with directions $(x, y, z)$. Equation (3.3) can be expanded out as follows

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho U_i U_j \right) = 0$$

(3.4)

The conservation of momentum likewise can be found in RANS notation to be

$$\frac{\partial}{\partial t} \left( \rho U_i \right) + \frac{\partial}{\partial x_j} \left( \rho U_i U_j \right) = -\frac{\partial \bar{P}}{\partial x_j} + \frac{\partial}{\partial x_i} \left( \mu \left( \frac{\partial U_i}{\partial x_j} \right) - \rho \overline{u_i u_j} \right)$$

(3.5)

where $\bar{P}$ is the mean pressure and the final term on the right of equation 3.5 in parenthesis is known as the stress tensor. The last term, $\overline{u_i u_j}$, is the Reynolds stress tensor which are non-zero for $i \neq j$.

### 3.3 Turbulence Model

This study uses a familiar type of turbulence models known as the linear eddy viscosity two-equation models which solve a set simultaneous transport equations for two turbulence quantities. A number of models were developed over the years from which it was chosen to proceed with the commonly known $k - \varepsilon$ model.

#### 3.3.1 Reynolds stresses

The emergence of the Reynolds stress term defines the influence of turbulence on the fluid motion. In its present form, $\overline{u_i u_j}$, it represents several unknown quantities that need to be solved for in order to solve or “close” the set of equations. This closure issue is dealt with by introducing a turbulence model which utilizes known quantities to approximate the Reynolds stresses. This study utilizes a linear eddy viscosity model (EVM) to take care of the closure issue by relating the Reynolds stresses linearly to the mean strains via a turbulent (eddy) viscosity. The linear EVM approximates the Reynolds stresses by way of the Boussinesq approximation which is written as

$$-\rho \overline{u_i u_j} = \nu_t \left( \frac{\partial \overline{U_i}}{\partial x_j} + \frac{\partial \overline{U_j}}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k$$

(3.6)
Where $\nu_t$ is the turbulent viscosity and is dependent on the fluid motion/turbulence rather than the fluid properties, $k$ is the turbulent kinetic energy, and $\delta_{ij}$ is the Kronecker delta and takes the value of 1 when $i = j$ and zero when $i \neq j$. $\nu_t$ is presumably defined by the properties of the turbulence since it describes the mean momentum transfer by the turbulent fluctuations.

The one issue with the Boussinesq approach is it relies on local isotropy where

$$\overline{u^2} = \overline{v^2} = \overline{w^2} = \frac{2}{3}k$$  \hspace{1cm} (3.7)

meaning the turbulence is distributed it evenly in each direction. The issue arises since turbulence is highly anisotropic and thus the model does not reflect that, especially in study like the present work. It is prudent to point out that this present study is therefore perhaps concerned with general reactions in the flow geometry rather than quantitative accuracy of the flow. However, quantitative comparisons with measurements will be made.

### 3.3.2 High-Re $k-\varepsilon$ model

A variety of models can be implemented to model the Reynolds stresses, the most common of which are the eddy viscosity models. These models draw a relation between the turbulent and molecular mixing as defined by equation 3.6. More specifically, the two-equation linear EVM was selected solving for the turbulence kinetic energy $k$ and the energy dissipation $\varepsilon$. In this study the EVM solves the kinetic energy transport equation given as

$$\frac{\partial k}{\partial t} = P_k - \varepsilon + \frac{\partial}{\partial x_i} \left[ (\nu + \nu_t) \frac{\partial k}{\partial x_i} \right]$$  \hspace{1cm} (3.8)

where $P_k$ is the energy production term or generation rate of turbulent kinetic energy, $\varepsilon$ is the dissipation rate of turbulent kinetic energy. The generation rate in an EVM is given by

$$P_k = -\rho \overline{u_i u_j} \frac{\partial \overline{u_j}}{\partial x_i}$$  \hspace{1cm} (3.9)

Substituting the Boussinesq approximation, from equation 3.6, in to model $P_k$ returns

$$P_k = \nu_t \left( \frac{\partial \overline{u_i}}{\partial x_i} + \frac{\partial \overline{u_j}}{\partial x_j} \right) \frac{\partial \overline{u_j}}{\partial x_i}$$  \hspace{1cm} (3.10)

The dissipation rate term is the dissipation rate of the turbulent kinetic energy per unit mass. The dissipation rate transport equation is derived by reference to the $k$ equation to be given as
Chapter 3: Governing Equations

\[
\frac{Dc}{Dt} = c_{e1} \frac{\varepsilon}{k} P_k - c_{e2} \frac{c}{k} + \frac{\partial}{\partial x_i} \left[ \left( \nu + \frac{\nu_t}{\sigma} \right) \frac{\partial \varepsilon}{\partial x_i} \right]
\]  

(3.11)

where \(c_{e1}, c_{e2},\) and \(\sigma\) are constants that have been tuned to a variety of flows. The empirical values for all the turbulence model constants can be found in table 3.1. The first term on the right is the source term which is define for if \(k\) is created by mean shear then the dissipation rate increases. The second term on the right is the sink term which is present to ensure that if \(P_k\) is zero then \(k\) and \(\varepsilon\) decrease. The high-Re \(k - \varepsilon\) model solves both equations (3.8, 3.11) simultaneously and defines the eddy-viscosity, \(\nu_t\), as

\[
\nu_t = c_{\mu} \frac{k^2}{\varepsilon}
\]

(3.10)

where \(c_{\mu}\) is a proportionality constant that is empirically defined. In equations 3.8 and 3.11 we see a \(\frac{D}{Dt}\) term denoting a total derivative of the form

\[
\frac{D\phi}{Dt} = \frac{\partial}{\partial t} (\phi) + \frac{\partial}{\partial x_i} (U_i \phi)
\]

(3.12)

where \(\phi\) is some variable and in the case of equations 3.8 and 3.11 the variables are \(k\) and \(\varepsilon\). Equation 3.12 represents the time dependent and convection terms for both transport equations. The very right term in both equations of interest denotes the diffusion term and both utilize an effective viscosity term defined by \(\nu + \frac{\nu_t}{\sigma}\) which includes both the kinematic and turbulent viscosities and the turbulent Prandtl number, \(\sigma\), for each respective transport equation. A lengthscale does not need to be prescribed as it comes out in the solution naturally as \(l = \frac{k^2}{\varepsilon}\).

Table 3.1: Empirically determined constants used in the high-Re \(k - \varepsilon\) turbulence model.

<table>
<thead>
<tr>
<th>(\sigma_k)</th>
<th>(c_{\mu})</th>
<th>(c_{e1})</th>
<th>(c_{e2})</th>
<th>(\sigma)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.0</td>
<td>1.44</td>
<td>1.92</td>
<td>1.3</td>
</tr>
</tbody>
</table>

It should be noted that in the far field the flow is dominated by the turbulent motion, but near the wall the flow is almost entirely influenced by the molecular viscosity in the region called the viscous affected sub-layer. These viscous effects are neglected in the \(k\)-epsilon turbulence model but accounted for by using a wall function across the viscous sub-region.
3.4 Near-Wall Region

The near-wall turbulent boundary layer is found to comprise of two regions, with the region closest to the wall being dominated by the molecular viscosity, fluid density, and wall shear stress, \( \tau_w \). As the flow reaches a certain distance away from the wall it transitions to a region dominated by the turbulent motion of the flow and becomes mostly if not completely independent of the fluid viscosity, but dependent of the fluid density, boundary layer thickness, distance from the wall, and wall shear stress. The inner region is a small portion of the boundary layer and split into yet two more distinct layers. These layers are commonly referred to as the viscous sub-layer (as noted earlier) and the turbulent sub-layer. In finite-volume methods quantities are approximated linearly across cells, but these approximations are not sufficient in the viscous sub-layer where steep wall-normal gradients are found unless a fine enough grid is used. Using a very fine mesh near the wall is acceptable, but three-dimensional flows can be very expensive as well as the near-wall cells can become elongated and present further problems with the stability of solution.

The widely-used alternative method is to employ a wall function to model the velocity profile across the boundary layer and obtain the corresponding shear stresses. The gradients in the directions along the wall can be neglected because they are typically much smaller than the wall-normal gradients. This leaves the total stress across both sub-layers to be defined as

\[
\tau = -\mu \frac{\partial U}{\partial y} - \rho \bar{u} \bar{v}
\]  

(3.13)

The viscous stress is denoted as \(-\mu \frac{\partial U}{\partial y}\) and the turbulent stress is defined as \(-\rho \bar{u} \bar{v}\). The total shear stress is assumed to be constant and equal to the wall shear stress across the inner region of the boundary layer.

![Figure 3.1: Inner sub-layers of near-wall boundary layer with streamwise velocity profile. Transition point found at point where velocity profile and viscous sub-layer curves meet.](image)
Chapter 3: Governing Equations

This transition point, denoted by $y^+$ in figure 3.1, is defined in non-dimensional distance

$$y^+ = \frac{y(\frac{\tau_w}{\rho})^{1/2}}{v} \quad (3.14)$$

where $y$ is the physical distance from the wall and transition to the turbulent sub-layer assumed at a $y^+$ of 11.2. This abrupt switch means in the viscous sub-layer $\tau_w = -\mu \frac{\partial U}{\partial y}$ and $-\rho \bar{u} \bar{v} = 0$, and similar for the turbulent sub-layer except $\tau_w = -\rho \bar{u} \bar{v}$ and $-\mu \frac{\partial U}{\partial y} = 0$. It is using this formulation that allows for a wall function such as the log-law to be used and a coarser mesh to be employed.

### 3.4.1 Log-law formulation

Log-law functions make the assumption the boundary layer is in local equilibrium where the increasing lengthscale is linearly proportional with the distance from the wall. As stated earlier, it can be assumed in fully-developed flow the velocity gradients are much larger in the wall-normal direction than either of the streamwise directions and thus these along with the stress gradients can be neglected. Looking at the updated streamwise momentum equation and integrating it twice gives

$$U = \left(\frac{\tau_w}{\mu}\right) y + \frac{y^2 \partial p}{2\mu \partial x} \quad (3.15)$$

where the pressure distribution is constant across the turbulent boundary layer. Equation 3.15 will only be an approximation for a portion of the viscous sub-layer when used a turbulent flow. The equation will not hold in the turbulent sub-layer and thus a lengthscale is of $l_m = \kappa y$ is used to reveal a log-law relation given as

$$U^+ = \frac{1}{\kappa} \log(Ey^+) \quad (3.16)$$

with $y^+$ defined as in equation 3.14 and

$$U^+ = U/(\tau_w/\rho)^{1/2} \quad (3.17)$$

Where $U^+$ is the non-dimensional velocity, $\kappa$ is the Von Karman constant, and $E$ is a simple integration constant. $\kappa$ and $E$ are usually taken to be 0.41 and 9 respectively. This wall function is appropriate for simple turbulent flows but it has weaknesses in the presence of adverse pressure gradients or any other situations where the wall shear stress goes to zero. From equation 3.13 one can see the wall shear stress goes to zero if...
these situations arise in the near-wall region, and in the case of the present study it means the turbulent viscosity disappears around the reattachment point in the near-wake behind the cylinder where turbulence has been proven to be quite active in this area (Karabelas 2010). Since this study is using the $k-\varepsilon$ two-equation model, values of the turbulence kinetic energy can be used to develop an improved log-law wall function.

### 3.4.2 Advanced Log-Law Function

The formulation of the log-law function using the turbulence kinetic energy stems from essentially the same idea where in an equilibrium boundary layer we will have

$$\overline{uv} = \frac{\tau_w}{\rho}$$  \hspace{1cm} (3.18)

$$\frac{\overline{uv}}{k} = c_{\mu}^{1/2}$$  \hspace{1cm} (3.19)

From equations 3.18 and 3.19 we have an expression for $k$ as

$$k = \left(\frac{\tau_w}{\rho}\right)c_{\mu}^{1/2}$$ \hspace{1cm} (3.20)

The above equation can then be used to prescribe a new non-dimensional velocity and wall distance by using $k^{1/2}$ in place of $(\tau_w/\rho)^{1/2}$. This substitution yields the following terms

$$U^* = U\frac{1}{\kappa^*}\left(\frac{\tau_w}{\rho}\right)^{1/2}$$  \hspace{1cm} (3.21)

$$y^* = y\frac{1}{\kappa^*}$$  \hspace{1cm} (3.22)

where $U^* = U\frac{1}{\kappa^*}c_{\mu}^{1/4}$ and $y^* = y\frac{1}{\kappa^*}c_{\mu}^{1/4}$ respectively and $c_{\mu}$ is the proportionality constant found in the turbulence model noted earlier (equation 3.10). The above expressions yield a modified log-law function given by

$$U^* = \frac{1}{\kappa^*}\log(E^*y^*)$$  \hspace{1cm} (3.23)

where $\kappa^* = \kappa c_{\mu}^{1/4}$ and $E^* = E c_{\mu}^{1/4}$. This formulation, as well as the previous formulation found in section 3.4.1, is solved iteratively as both are implicit functions. The function can be solved over a sufficiently large near-wall cell which lies outside the viscous sub-layer but inside the fully turbulent sub-layer, meaning a very fine near-wall mesh is no longer necessary. Using the known velocity, wall distance, and density at the node closest to the wall, one can determine the wall shear stress. This advanced form of the log-law function is widely-used in modern software packages because it reasonably approximates flow in the viscous sub-layer and accurately describes the flow.
Chapter 3: Governing Equations

in the turbulent sub-layer. To find the effective viscosity, equation 3.23 is integrated once with respect to \( y \) yielding

\[
\frac{\partial u}{\partial y} = \frac{k^{1/2}}{\kappa y}
\]  

(3.24)

Using equation 3.13 and the assumption that the total shear stress is equal to the wall shear stress across the inner region of the turbulent boundary layer, the effective viscosity can be written as

\[
\tau = \tau_w \text{ thus } \mu_t = \rho \kappa y k^{1/2}
\]  

(3.25)

This above expression for effective viscosity does not disappear when the wall shear stress equals zero. With this wall function the turbulence at and around near-wake reconnection point in the flow behind the cylinder can be solved. In laminar flow the reconnection point usually lies outside the near-wall region but in high-Re flows this point moves closer to the cylinder as shown in Breuer 2000.

3.5 Summary

This chapter defined and outlined the governing equations solved through approximation and iterative techniques. The chapter started by defining the widely-known Navier-Stokes equations, but these equations on their own are very complex and take quite a long time to solve. The Reynolds-averaged Navier-Stokes equations were introduced to simplify the task by expressing the original equations in terms of averaged quantities and their fluctuating components. The fluctuating components are known as the turbulent fluctuations and are negligible in a laminar flow. The present work models a turbulent flow over the Flettner rotor and thus these fluctuating components must be solved for and the study uses a widely-used turbulence model to solve them. The high-Re \( k - \varepsilon \) turbulence model used in the present study was described. The modelled Reynolds stresses are broken down and a description of the turbulence motion was described. The RANS equations with the high-Re \( k - \varepsilon \) model will accurately describe the flow behaviour for the entire solution space except for the very near-wall region. Since building a mesh which is very fine near the wall to accurately model the viscous effects and shear stresses is quite expensive, a coarser mesh is used where just one node within the near-wall region is present and a wall function is used to model the non-linear velocity profile near the wall. The next section defines the common log-law
wall function and then describes the advanced log-law formulation used in the present study which acts as a damping function for the molecular viscosity in the viscous sub-layer, but still applies to the turbulent sub-layer where viscosity is no longer reliant on the fluid properties. This allows the near-wall node to be placed in the fully turbulent region of the boundary layer and an iterative method used to solve for the near-wall shear stress and effective viscosity, a calculated viscosity used across both inner sub-layers. A discretization technique must be applied to this governing set of equations to solve the equations in an approximate but accurate way which is the subject of chapter 4.
Chapter 4: Numerical Methods

4 Numerical Methods

4.1 Discretization Scheme

The finite volume method is adopted in this study which essentially breaks down the solution space into a finite number of control volumes. The governing equations are integrated over each control volume to give way to the resulting integrated transport equations. The transport equations are then discretized further using a variety of the discretization schemes. In this study, the resulting convective terms of the momentum equations are discretized using a variation of the QUICK scheme (defined later) while the turbulence quantities are discretized with the first-order upwind differencing scheme to avoid potential periodic oscillations caused by the steep turbulence gradients commonly found in the near-wake region of the cylinder. The diffusion terms are, however, discretized using the central difference scheme. The discretized equations are then solved iteratively. A model one-dimensional transport equation to display the procedure is given as follows

\[
\frac{\partial (\rho U \phi)}{\partial x} = \frac{\partial}{\partial x} \left( \kappa_d \frac{\partial \phi}{\partial x} \right) + S_{cv}
\]  

(4.1)

where \( \phi \) is some transported variable, \( \kappa_d \) is the respective diffusion coefficient, and \( S_{cv} \) is the collective source terms. Taking the integral over a single control volume yields

\[
\int_\Omega \frac{\partial (\rho U \phi)}{\partial x} \, d\Omega = \int_\Omega \frac{\partial}{\partial x} \left( \kappa_d \frac{\partial \phi}{\partial x} \right) \, d\Omega + \int_\Omega S_{cv} \, d\Omega
\]  

(4.2)

where \( \Omega \) is the cell volume. Seeing this is a 1-D example, the integration from the west to the east faces is desired and the cross-sectional area, \( A \), assumed to be known and constant. The resulting expression becomes

\[
(\rho U A \phi)_e - (\rho U A \phi)_w = \left( \kappa_d A \frac{\partial \phi}{\partial x} \right)_e - \left( \kappa_d A \frac{\partial \phi}{\partial x} \right)_w + S_{cv}^P \Delta \Omega
\]  

(4.3)

where the lower case subscripts \( e, w \) denote the values at the midpoints of the east and west faces of the cell, and the term \( S_{cv}^P \) denotes the source terms at node P (the centre of the cell). With the convection terms needing no further simplification, the diffusion terms can be approximated linearly using a central differencing discretization scheme revealing diffusion terms expressed in terms of nodal values at the east and west faces.

\[
\left( \kappa_d A \frac{\partial \phi}{\partial x} \right)_e = \kappa_d A \frac{\phi_e - \phi_P}{\Delta x_P}
\]  

(4.4)
Chapter 4: Numerical Methods

\[
\left( \kappa_d A \frac{\partial \phi}{\partial x} \right)_w = \kappa_d A \left( \frac{\phi_p - \phi_W}{\Delta x_{PW}} \right) \tag{4.5}
\]

with the labels P, E, and W pointing to the nodal values of the current cell and the neighbouring east and west cells. The term \(\Delta x_{PE}\) denotes the distance along the line from node P to node E. Substitute expressions 4.4 and 4.5 into equation 4.3 and the equation becomes

\[
(\rho UA \phi)_e - (\rho UA \phi)_w = \kappa_d A \left( \frac{\phi_E - \phi_P}{\Delta x_{PE}} \right) - \kappa_d A \left( \frac{\phi_P - \phi_W}{\Delta x_{PW}} \right) + S^P \Delta \Omega \tag{4.6}
\]

Consolidating constants to simplify equation 4.6 to

\[
F_e \phi - F_w \phi = D_e (\phi_E - \phi_P) - D_w (\phi_P - \phi_W) + S^P \Delta \Omega \tag{4.7}
\]

Where the mass flux through the cell face is denoted \(F = \rho UA\) and \(D = \kappa A / \Delta x\) is the collective diffusion coefficient.

4.2 Convection Scheme

The convective fluxes are solved one of two ways depending on the transport variable being solved. The convective terms of the momentum transport equations are discretized using a higher-order variation of the QUICK (quadratic upstream weighted interpolation for convective kinematic) scheme which also incorporates a ‘deferred correction’ scheme. The turbulent convective terms are solved using an upwind differencing scheme. Convective flow behaviour is often measured by defining a dimensionless quantity known as the Peclet number. The Peclet number in regards to mass flow is given by

\[
P_e = \frac{d \text{U}}{\gamma} \tag{4.8}
\]

where \(d\) is the characteristic length, in this case the cylinder diameter, \(U\) is the streamwise uniform velocity, and \(\gamma\) is the mass diffusion coefficient. The Peclet number is a non-dimensional quantity that measures the ratio of convection to diffusion. The higher-order scheme used in this study is organically the QUICK scheme, but these types of schemes are susceptible to artificial oscillations when the Peclet is high and steep gradients are present in the flow. The variation of the QUICK scheme used in the present study incorporates a sort of deferred correction to dampen these effects. A first-order correction scheme will undoubtedly yield stable results but lacks accuracy where a higher-order scheme (higher than 2\(^{nd}\) order) will increase the accuracy but not guarantee
stability in every flow situation. The alternative and seemingly optimum route would be a hybrid scheme of a higher-order with an oscillation damping component.

### 4.2.1 UMIST scheme

The convection scheme used in the present study is the Upstream Monotonic Interpolation for Scalar Transport (UMIST) scheme from Lien and Leschziner (Upstream, 1994). This is a scheme developed by the same individuals who designed the STREAM code being used in this present study (Lien & Leschziner; 2nd-Mom Turb-Trans, 1994). The scheme aims to apply a higher-order approximation without generating the artificial oscillations when steep gradients are present and the Peclet number, $Pe$, is sufficiently high. The scheme diffuses the convective effects by placing greater bias toward the upstream nodes while keeping the order of accuracy, or preserving monotonicity in the scheme. In order to achieve monotonicity, the upstream biasing must be controlled by the oscillatory features of the solution (Lien and Leschziner; Upstream, 1994). For the scheme to achieve this it must be non-linear.

The oscillation limiting concept of Total Variation Diminishing was theorized by Harten (1983) to develop high-resolution convection schemes which combine accuracy with monotonicity and entropy preservation (Lien & Leschziner, Upstream 1994). Through the flowing years, many TVD diagrams were developed based on the QUICK scheme but were very computationally intensive as they used many conditional statements in the algorithms. Lien and Leschziner were able to develop a highly compact yet continuous QUICK-based limiter for a non-orthogonal collocated finite-volume solution known as the UMIST scheme.

The UMIST scheme usually demands about 20% more CPU time than the traditional QUICK scheme, but the additional time is acceptable since a more stable solution is obtained.

### 4.3 Time Discretization

The cases in this study are unsteady in nature which requires the time derivative in the governing equations to be discretized. The Crank-Nicolson method is used to approximate the time derivative of the RANS equations. First the generic time-dependent problem below is considered
\[ \frac{\partial \phi}{\partial t} = f(t, \phi(t)) \]  

(4.31)

where \( \phi \) is some transport variable and \( f \) is some given function. The time discretization is accomplished by taking the integral of the above equation over one time step, thusly expressed as

\[ \int_t^{t+\Delta t} \frac{\partial \phi}{\partial t} \, dt = \int_t^{t+\Delta t} f(t, \phi(t)) \, dt \]  

(4.32)

rewritten as

\[ \phi^{n+1} = \phi^n + \int_t^{t+\Delta t} f(t, \phi(t)) \, dt \]  

(4.33)

where \( n \) is the superscript denoting the quantity evaluated at time \( t \), with \( (n + 1) \) at time \( t + \Delta t \). To understand the Crank-Nicolson scheme, we look at the time-dependent convection/diffusion problem

\[ \frac{\partial \phi}{\partial t} + \frac{\partial}{\partial x_j} (U_j \phi) = \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial \phi}{\partial x_j} \right) \]  

(4.34)

Using equation 4.33 in a 1D temporal discretization of the above equation and central differencing for the convection and diffusion terms, one approximation of the time discretization is the explicit Euler scheme defined as

\[ \phi_i^{(n+1)} = \phi_i^{(n)} + \Delta t \left[ -U_i \left( \frac{\phi_i^{(n)} - \phi_{i-1}^{(n)}}{2\Delta x} \right) + \Gamma \left( \frac{\phi_i^{(n)} - 2\phi_i^{(n)} + \phi_{i+1}^{(n)}}{(\Delta x)^2} \right) \right] \]  

(4.35)

Explicit schemes generally have problems with stability and need a very small time step to achieve stability. The Crank-Nicolson scheme is a generic scheme that utilizes the trapezium rule and results in a second order accurate solution. Written in a formula similar to that of equation 4.33 and 4.35, the Crank-Nicolson scheme is

\[ \phi_i^{(n+1)} = \phi_i^{(n)} - \frac{U_i \Delta t}{2} \left( \frac{\phi_{i+1}^{(n+1)} - \phi_{i-1}^{(n+1)}}{2\Delta x} \right) + \frac{\Gamma \Delta t}{2} \left( \frac{\phi_{i+1}^{(n+1)} - 2\phi_i^{(n+1)} + \phi_{i-1}^{(n+1)}}{(\Delta x)^2} \right) \] \[- \frac{U_i \Delta t}{2} \left( \frac{\phi_{i+1}^{(n)} - \phi_{i-1}^{(n)}}{2\Delta x} \right) + \frac{\Gamma \Delta t}{2} \left( \frac{\phi_{i+1}^{(n)} - 2\phi_i^{(n)} + \phi_{i-1}^{(n)}}{(\Delta x)^2} \right) \]  

(4.36)

The scheme is generally unconditionally stable via the Von-Neumann analysis but has proven to produce instability for large time steps. That being said, the Crank-Nicolson scheme has the ability to take larger time steps due to its second-order accuracy compared to first-order schemes, while retaining its temporal accuracy.

### 4.4 Boundary Conditions
The boundary conditions determine the flow structure on the extremities of the solution space as well as any walls defined within the solution space. The boundary conditions are applied with educated expectations of how the flow should behave thus their application is aimed to reduce computation time and provide accurate results.

4.4.1 Fluid Inlet

The inlet employs Dirichlet conditions to all variables solved by the governing transport equations. The inlet boundary is curved to contour the cylinder where \( U = 1 \) and \( V = 0 \) ensuring there is a non-zero component normal to the boundary. Conditions for the pressure are not needed when a prescribed velocity is defined. The inlet turbulence quantities were set with the basic turbulence kinetic energy equation \( k = (IU)^2 \), where \( U \) is the inlet velocity set as noted above and \( I \) is the turbulence intensity taken to be 1% at the inlet.

4.4.2 Wall

The cylinder wall and the disc walls (as in the cases with Thom discs) are applied no-slip and impermeability conditions. These conditions are difficult to solve with the applied wall function and the curved surfaces found in the present study. The wall shear stress is defined and solved for with the log-law wall function. The turbulent energy production term \( P_k \) and dissipation term \( \varepsilon \) at the near wall cell are solved in a way which is consistent with the log-law function defined in section 3.4.2. Further details of the wall conditions can be found in Lien & Leschziner (1994, 2nd-Mom Turb-Trans).

4.4.3 Fluid Exit

The solution domain applies zero gradient conditions at the fluid exit. The exit employs an inlet/outlet mass conservation by way of a usually very small pressure adjustment applied across the last half each cell at the fluid exit.

4.4.4 Periodic boundary

Periodic conditions are applied to the 3D simulations along the spanwise direction of the solution domain. The specific faces these conditions are applied are the cylinder end faces (i.e. \( z = 0 \) and \( z = Z_{max} \); these dimensions become clear in later chapters) In the cases where Thom discs are present, this condition approximates interactions from
cavity to cavity, however, Craft et al. (2010) hinted this type of condition in its present use may not be physical and a domain incorporating the entire rotor could be the only way to sufficiently model these types of flows.
5 Axial Effects on Rotating Smooth Cylinder Flows

5.1 Preliminary Remarks

Rotating cylinder flows in two-dimensions have been a primary focus of many past numerical investigations. Many of these studies focused on the near-wake behaviour and the modification and eventual suppression of the Von Karman vortex street. Investigations in two-dimensions are, arguably, suitable for laminar flows but when the flow transitions into the turbulent regime, it produces small turbulence structures and becomes three-dimensional. A handful of numerical investigations are relevant to the flows examined in the present 3D study. The most notable are those of El Akoury et al. (2009) who looked at very low-Re rotating cylinder flows in 2D and 3D for $\alpha$ up to 6, Breuer (2000) who studied high-Re flows over a stationary cylinder in 3D with variations in cylinder span up to $2\pi R$ ($R$ is defined as the radius of the cylinder), Karabelas (2010) who examined high-Re rotating cylinder flows in 3D for $\alpha \leq 2$, and lastly Craft et al. (2010, 2011) who carried out a 3D high-Re studies for rotation rates up to 8. The present chapter aims to compare the results obtained in the present study with those reported by the studies noted above to confirm the numerical methods and discuss the effects of cylinder span length on the individual flows.

The present study is a 3D numerical investigation of the flow over a rotating smooth cylinder (figure 5.1). Simulations for $Re = 140,000$ were examined. Although the Reynolds number is not representative of actual Flettner rotor operating conditions, it is, however, sufficiently high to place the flow well into the turbulent flow regime and corresponds with that of a major LES study (Karabelas, 2010). The distinguishing characteristics of each simulation are the dimensionless spin rate, $\alpha$, and the dimensionless cylinder span length, $Z_{\text{max}}/R$. The study simulates flows for $\alpha = 2$ and 5 and $Z_{\text{max}}/R = 2$ and 6; four simulations in total. The axial span lengths of the computational domains in the present study were chosen to coincide with Breuer (2000) who used span lengths $2R$ and $2\pi R$. The shorter span length was also comparable to that of Karabelas (2010) and Craft et al. (2010, 2011). The main aim of these
simulations is to determine the effects of the aerodynamic performance on the axial length considered.

### 5.2 Grid Resolution

The three-dimensional grid was designed to capture the structure of the flow by emphasising the near-wall and near-wake regions around the cylinder. The grid used in this study was a C-type mesh where the inlet is contoured to the cylinder and the mesh opens up to a rectangular cross-section to ensure the downstream flow is accurately resolved. The 2D profile of the grid is shown in figure 5.2 which highlights the large concentration of nodes near the cylinder wall. The near-wall mesh (figure 5.3) was designed using the log-law wall function discussed in section 3.4. Optimally, the near-wall cells were to be placed in the fully turbulent near-wall layer of the boundary layer which was accomplished by calculating the non-dimensional $y^+$ values by a simple guess and check procedure. The near-wall mesh was considered optimized once at least 80% of the near-wall cells returned $y^+ > 30$. The other 20% of the cells were usually located in the recirculation region behind the cylinder and commonly returned low values of $y^+$ since the friction velocity contribution was very small in these areas. Refer to section 3.4 for further information regarding $y^+$ and the log-law formulation.

The mesh concentration along the cylinder span (z-direction) is uniform, meaning there is no bias toward either end or the centre of the cylinder span. The spanwise spacing of nodes was based on the earlier studies at Manchester where the density of nodes had been systematically examined to capture the large-scale motion along the cylinder. Figure 5.3 shows the 2R-SC and 6R-SC grids in 3D where the cylinder wall is highlighted in purple. (Refer to table 5.1 below for grid size and key dimensions.)

<table>
<thead>
<tr>
<th>Grid</th>
<th>Total blocks</th>
<th>Total nodes</th>
<th>Near-wall nodes</th>
<th>Spanwise nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>2R-SC</td>
<td>17</td>
<td>526,368</td>
<td>3,968</td>
<td>32</td>
</tr>
<tr>
<td>6R-SC</td>
<td>17</td>
<td>690,858</td>
<td>5,208</td>
<td>42</td>
</tr>
</tbody>
</table>

Grid resolution will have convergence issues if it is too coarse in the near-wall regions as these are the areas where the largest gradients in the velocity and pressure
fields lie. A coarse near-wall mesh may result in periodic oscillations from node to node. The grid resolutions in the present study were largely based on previous studies at Manchester and simply were modified versions of known converging grids similar to those of Craft et al (2010, 2011).

5.3 Axial Effects of Flow Structure

The axial effects of the flow structure past smooth cylinders were examined for a relatively low spin rate and a higher rate to be within the range of rotational speeds experienced by Flettner-rotor crafts. The 3D simulations were then aerodynamically compared with 2D computations and select results of previous studies.

5.3.1 Mean Near-Wake Profile Structure

A comparison of the mean flow structure with Karabelas (2010) for $\alpha = 2$ is shown in figure 5.4. The present computations exhibited good agreement for the shape of the near-wake region compared with Karabelas (2010). The overall magnitude of $k$ found by Karabelas (2010) was sufficiently higher. The difference in magnitude arises from the fact that the present study has not included the effective turbulence energy associated with the time-dependent fluctuations of the “mean” velocity field (an omission that was recognized only after the instantaneous data had been processed and discarded). Karabelas (2010) solved the total resolved kinetic energy which largely consisted of the contribution of the fluctuating kinetic energy component. As noted, this contribution was not included in the present study given its qualitative emphasis and thus only the turbulence energy solved for in the model is shown.

5.3.2 Low Rotation Span Length Dependency

Briefly mentioned earlier, the computational domains have similar defined span lengths to that of Breuer (2000). Breuer (2000) concluded that no significant effects could be noticed with increases in cylinder span, but his cases considered only flow past non-rotating cylinders and no direct relation can be made for rotating cylinder flows.

For $\alpha = 2$, span lengths 2R and 6R were compared and the flow structure examined. The velocity fields shown in figure 5.5 exhibited similar recirculation
regions in the near-wake. These were consistent with the previous URANS study at Manchester by Craft et al. (2011). The purely 2D computation exhibited similar velocity field with the fluid separation over upper side slightly further upstream than those exhibited by the 3D results. Comparing these results with those of Karabelas (2010), the LES data showed the lower recirculation region completely suppressed. The differences were judged to be acceptably small being a feature of the different computation methods employed (i.e. present URANS versus Karabelas’ (2010) LES). In fact, the present results showed very small perturbations for both 2R and 6R span lengths. The perturbations (in lift and drag) agree with the velocity fields, but were far too small to be considered vortex shedding. Little or no spanwise motion was found for either span length arguably indicating the flow with relatively low spin rate is essentially two-dimensional.

5.3.3 High Rotation Span Length Dependency

The increased rotation rate, $\alpha = 5$, created much increased complexity in the flow. The two span lengths saw similar streamwise motion indicated by the velocity fields in figure 5.6. Let it be noted the figure presents velocity field with 3D streaklines projected on a 2D frame of reference. The increase in spin rate $\alpha$ produced 3D structures, discussed later in this section, in the downstream wake (see 3D streaklines in figure 5.11) exhibiting the appearance of the streaklines crossing. Turning attention to the instantaneous spanwise motion, both span lengths saw similar undulation around the cylinder. Figures 5.8 and 5.9 show the spanwise motion along the cylinder for various planes normal to the cylinder at geometric positions corresponding to the diagram in figure 5.7. The 2R span exhibited a larger magnitude in the spanwise component of velocity ($W/U$) compared to the 6R span, but showed a weakly developed structure along the span length where the 6R span saw more distinct structures. It is believed the periodic boundary condition applied on the top and bottom cylinder ends may have caused some unphysical spanwise motion for the 2R span simulation.

The greatest magnitude of spanwise motion was found on the upper side of the cylinder (position A). Focusing on the 6R span, the temporal evolution of the spanwise motion is shown in figure 5.10. The development of the spanwise undulation in this simulation started on the ends and worked toward the centre of the span. The motion exhibited little structural change from $40 \leq t \leq 100$ with small discrepancies notably due
to the turbulence of the opposing flows over the upper side of the cylinder. The structured undulation over the upper side of the cylinder with the longer span was investigated further. Spanwise trailing vortices were found and shown in figure 5.11. The vortices along the span correspond to the spanwise motion shown in figure 5.10. These vortices were dependent on spin rate as they were not found for $\alpha = 2$. The shorter span exhibited similar behaviour but here the individual vortices interacted downstream of the cylinder where the longer span did not. The behaviour may be a result of the periodic boundary condition mentioned earlier.

Looking back to figures 5.8 and 5.9, the spanwise motion around the cylinder suggested some spanwise circulation was found in the near-wall region around the entire cylinder for both span lengths examined.

### 5.4 Axial Effects of Aerodynamic Performance

The aerodynamics of a rotating cylinder has been widely reported for laminar and turbulent flows (Prandtl 1925, Thom 1934, Mittal & Kumar 2003, Karabelas 2010). The rotor in the present study rotated counter-clockwise with flow moving from the left to generate a downward lift via the Magnus effect. The aerodynamic results were compared with present 2D computations and results from comparable studies.

The pressure distributions around the cylinder wall are shown in figure 5.12. The reference pressure was given for $C_p = 1$ at $\theta = 0$ with $\alpha = 0$, where the stagnation point translates accordingly with increasing $\alpha$. The pressure variations for $\alpha = 2$ exhibit no significant differences with the increase in span and thus led to very close agreement for the values of $C_l$ and $C_d$ shown in table 5.2. The lift and drag for $\alpha = 2$ in table 5.2 exhibited results with little or no dependency on span length. Karabelas (2010) and the 2D computation exhibited higher lift coefficients as can be found in the comparison generated in figure 5.13. The discrepancy with Karabelas (2010) was acceptable given the difference in the numerical modelling method.

Small discrepancies were found in both the lift and drag coefficients for the present $\alpha = 5$ simulations with the 6R span exhibited approximately 3% greater lift and lower drag than the 2R span. The temporal evolution of the lift coefficients again showed that the corresponding 2D computation exhibited higher lift but this relationship between 3D and 2D computations is qualitatively consistent with the results of the 2D
numerical and experimental comparison found in Aoki & Ito (2001). The temporal evolution of the lift coefficients can be found in figure 5.14. It is worth noting the original expectations were for the lift to decrease with increased span length given the idea that a 2D computation exhibits higher lift as it essentially simulates a span length of zero. Craft et al. (2011) found higher lift for $\alpha = 5$, yet not substantially higher considering their study was for $Re = 800,000$ thus indicating that spin ratio has a greater effect on the lift than the Reynolds number (based on the uniform free stream velocity). Returning to figure 5.12, the pressure distributions around the cylinder for each case mentioned above were consistent with table 5.1.

The temporal evolution of the drag coefficient is shown in figure 5.15. For both rotation rates, the evolution of the 2R span showed greater unsteadiness than the 6R span. As expected, given the results of the motion for both rotations, the higher rotation rate exhibited greater unsteadiness compared to the lower rotation rate. The temporal evolution of both coefficients for the higher spin ratio exhibited unsteadiness in the flow when the initial simulations up to $t = 100$ were analysed. These simulations were then expanded out to $t = 200$ where the slight, seemingly random, unsteady oscillations were found to continue with increasing time. From the information provided by this study, it can be hypothesised that the three-dimensionality of a rotating cylinder flow creates increasing unsteadiness as the rotation rate, $\alpha$, increases. Further research with regards to 3D smooth cylinder flow simulations should be conducted to confirm it is a result of the rotation and not a modelling related error.

<table>
<thead>
<tr>
<th>Dim.</th>
<th>Span</th>
<th>$\alpha$</th>
<th>total Cl</th>
<th>total Cd</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D</td>
<td>-</td>
<td>2</td>
<td>-5.6828</td>
<td>0.1500</td>
</tr>
<tr>
<td>3D</td>
<td>2R</td>
<td>2</td>
<td>-4.7985</td>
<td>0.1858</td>
</tr>
<tr>
<td>3D</td>
<td>6R</td>
<td>2</td>
<td>-4.7972</td>
<td>0.1877</td>
</tr>
<tr>
<td>Karabelas</td>
<td>2R</td>
<td>2</td>
<td>$\approx$-5.259</td>
<td>$\approx$0.128</td>
</tr>
<tr>
<td>2D</td>
<td>-</td>
<td>5</td>
<td>-12.6954</td>
<td>0.2239</td>
</tr>
<tr>
<td>3D</td>
<td>2R</td>
<td>5</td>
<td>-11.1311</td>
<td>0.1760</td>
</tr>
<tr>
<td>3D</td>
<td>6R</td>
<td>5</td>
<td>-11.4439</td>
<td>0.1858</td>
</tr>
<tr>
<td>Craft et al.</td>
<td>2R</td>
<td>5</td>
<td>$\approx$-12.1512</td>
<td>-</td>
</tr>
</tbody>
</table>
5.5 Summary of Findings

The main aim of this chapter has been on establishing what effect an increase in span length would have on a rotating cylinder flow. Breuer (2000) found a variation in span length did not have a large effect on the non-rotating flow studied. The initial assumption made by Karabelas (2010) was that the results from Breuer (2000) would hold for rotating cylinder flows and it was one of this study’s objectives to determine if this assumption was correct.

The results for the cases of the lower rotation rate of $\alpha = 2$ yield virtually identical results for the two span lengths. The fluid motion and the rotor aerodynamics exhibited little or no discrepancies with increased axial length. The present study returned aerodynamically weaker lift coefficients than those of Karabelas (2010), yet the relatively simple eddy viscosity model employed in the present study probably achieved sufficiently accurate results for many engineering applications. The increase in spin rate to $\alpha = 5$ saw some small differences in flow for the two axial lengths. Spanwise undulations were found for both span lengths which coincided with circulation found in the near-wall region over the upper side of the cylinder. The near-wall circulation translated into trailing vortices under the influence of the uniform flow. The shorter span captures these vortices but is believed the periodic boundary condition on the axial ends of the cylinder may have led to some unphysical interactions between the vortices downstream which wasn’t found with the $Z_{\text{max}} = 6R$ simulation.

The computations for $\alpha = 5$ exhibited small discrepancies in the aerodynamic properties between the two span lengths. The larger span exhibited higher lift and lower drag than the smaller span, but as mentioned these differences were very small. Overall, the results from the present study, there was no significant effects for an increased axial length within the solution domain.
5.6 Figures

Figure 5.2: (left) A single x-y plane cross-section of 3D smooth cylinder mesh. (right) Near-wall mesh created using $y^+$ and employed the log-law wall function.

Figure 5.3: (left) 3D C-type smooth cylinder mesh with $Z_{max} = 2R$, note the cylinder surface is highlighted in purple. (right) 3D C-type smooth cylinder mesh with $Z_{max} = 6R$, note the cylinder surface is highlighted in purple.
Figure 5.4: Mean turbulence kinetic energy near-wake profiles compared with Karabelas (2010).
Chapter 5: Axial Effects on Rotating Smooth Cylinder Flows

Figure 5.5: Mean velocity field streaklines for $\alpha = 2$, compared with Karabelas (2010) and 2D computations.

Figure 5.6: Mean velocity field streaklines for $\alpha = 5$ compared with 2D computations. Refer to section 5.3.3 for explanation regarding the appearance of crossed streaklines.
Figure 5.7: 2D diagram of slices made through the cylinder at given y values. Flow is left to right, diagram corresponds to successive figures.
Figure 5.8: Streamwise contours of $W/U$ for $\alpha = 5$ and $Z_{max}/R = 2$ at $t = 100$. Flow is left to right, labels on left corresponds to diagram in figure 5.6.
Figure 5.9: Streamwise contours of W/U for α = 5 and Z_{max}/R = 6 at t = 100. Flow is left to right, labels on left corresponds to slices in figure 5.6.
Figure 5.10: Temporal development of streamwise contours of $W/U$ for $a = 5$ and $Z_{max}/R = 6$, planes corresponds to position A in figure 5.6. Flow comes from left, labels on left correspond to non-dimensional time, $t$. 
Chapter 5: Axial Effects on Rotating Smooth Cylinder Flows

Figure 5.11: 3D velocity field streaklines with vorticity contouring of the flow over the cylinder (using the orientation in fig. 5.1, \( \omega = \alpha \)).

Figure 5.12: Distribution of the pressure coefficient for all smooth cylinder cases in the present study, also includes purely 2D equivalent cases and results from Karabelas (2010) (\( \theta \) measured according to figure 5.1).
Chapter 5: Axial Effects on Rotating Smooth Cylinder Flows

Figure 5.13: Lift coefficient comparison as a function of spin ratio is graphically displayed. Present model computational results are given as data points due to how few of each case were simulated. Lines connecting these points would not best represent the lift coefficient behaviour with spin ratio.

Figure 5.14: Temporal evolution of the coefficient of lift for the present 3D smooth cylinders cases, the corresponding 2D cases, and a \( \text{Re} = 800k \) with \( \alpha = 5 \) from Craft et al. (2011).
Figure 5.15: Temporal evolution of the coefficient of drag for the present 3D smooth cylinders cases and the corresponding 2D cases.
Chapter 6: Flettner Rotor Flow with Thom Discs

6 Flettner Rotor Flows with Thom Discs

6.1 Preliminary Remarks

Thom (1934) speculated that the Magnus lift force could be improved and surpass Prandtl’s (1925) proposed limit by fixing evenly-spaced circumferential discs on the cylinder. The rationale was that the discs would reduce the amount of motion along the cylinder span and consequently reduce the drag. Indeed, Thom’s experiments recorded data for Flettner rotor flows with added discs for Reynolds number in the range $5200 < \text{Re} < 12,500$ for a number of disc sizes and spacing. For flows with larger disc diameters higher lift coefficients were indeed recorded at high spin rates than found by Prandtl (1925). The numerical study by Craft et al. (2010, 2011) examined a Flettner rotor flow with smaller discs and the distance between adjacent discs larger than those experimentally examined by Thom (1934). They found only marginal increase in lift coefficient for dimensionless rotation rates greater than 3. The present study has therefore, further examined the effects of adding discs but for disc diameter ratios and spacings comparable with those examined by Thom (1934).

The numerical implementation solves a single disc cavity section along the cylinder with infinitesimally thin discs. The present study employs two simulations at $\text{Re} = 140,000$ with $\alpha = 5$ and the disc spacing $Z_{\text{disc}}/R = 1$. All the figures show the Flettner rotor spinning in a counter-clockwise direction with a uniform flow coming from the left of the solution space resulting in a downward lift force. The study examines the performance of discs of diameter $D/R = 4$ and $D/R = 6$. A brief description of the flow structure within the cavity will be attempted as will the flow stability. Most attention is directed at the aerodynamic performance with comparison to previous Flettner rotor studies at Manchester and the present smooth cylinder flows (i.e. without discs). The presence of the discs greatly increases the complexity of the near-cylinder flow.
6.2 Grid resolution

The grid structure used in these Thom disc simulations is quite similar to that of the smooth cylinder simulations. The C-type grid is adopted where the internal boundary matches the contour of the cylinder and the mesh right of the rotor extends into a rectangular cross-section at the fluid exit. Wall boundary conditions via wall functions are applied to the cylinder surface and disc surfaces within the inner 4 blocks, highlighted in purple in figure 6.2. The Thom discs are modelled by applying the wall boundary conditions to the outer-most layer (on the cylinder ends) of nodes (inner four blocks) giving the discs an infinitesimally small thickness. The near-wall node spacing in both grids shown in figure 6.1 steadily reduces towards the cylinder wall. The spanwise nodal distribution employs a higher concentration of nodes at each end to model the near-wall motions close to the Thom discs. The z-direction clustering then reduces with radial distance from the cylinder and is eventually eliminated at the far edges of the solution. For each wall the near-wall cell is placed within the fully turbulent layer. The velocity gradient at the near wall node is modelled using the log-law wall function formulation described in section 3.4.2. The discs are modelled by adjusting the geometry of the inner four blocks to match the intended physical geometry. The radial extension of the inner four blocks found in the 6R-TD grid (figure 6.2) is compensated for in the neighbouring blocks. (Refer to table 6.1 below for grid size and key dimensions.)

<table>
<thead>
<tr>
<th>Grid</th>
<th>total blocks</th>
<th>total nodes</th>
<th>near-wall nodes</th>
<th>spanwise nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>4R-TD</td>
<td>17</td>
<td>526368</td>
<td>6200</td>
<td>32</td>
</tr>
<tr>
<td>6R-TD</td>
<td>17</td>
<td>548768</td>
<td>6820</td>
<td>32</td>
</tr>
</tbody>
</table>

6.3 Thom Disc Affected Flow Behaviour

Here the flow behavioural aspects of two simulations of a Flettner rotor with Thom discs are examined. The governing parameters of rotor flow used by Thom discs used by Thom (1934) were the ratio of the disc to cylinder diameters and the normalized
Chapter 6: Flettner Rotor Flow with Thom Discs

distance between the discs. The present study found the flow pattern was effectively
governed by the dimensions of the Thom disc cavity defined by the rectangular cross-
section between adjacent Thom discs and cylinder wall. The parameters of the Thom
disc cavity are the distance between the discs $Z_{\text{disc}}$ and the diameter of the discs $D$
normalized with the cylinder radius $R$. The Flettner rotor geometry used in this chapter
is shown in figure 6.3. In the present study $Z_{\text{disc}}$ is fixed at $1R$ and the simulations of
$D/R = 4$ and 6 are investigated with emphasis on a qualitative discussion of the flow
behaviour as it relates to the flow stability. The idea Thom discs develop higher lift and
lower drag (in relation to a rotating smooth cylinder) will be discussed.

6.3.1 Temporal Changes in Flow Past the Flettner Rotor

The present flows are very complex which makes it difficult to provide an explanation
for the unsteadiness found. Using the temporal evolution plot in figure 6.4, significant
unsteadiness for both simulations was found in the temporal region from approximately
$150 < t < 175$. The plot illustrates the randomness found within a single oscillation
period. The shapes of the curves in the plot are similar in periodic nature, but notably
the two simulations are different and the similarity in the curves is fortuitous. The
labelled positions (A-E) in time were selected to bring out the varied flow structures
that result in the corresponding magnitude of lift. Several figures in this chapter refer to
the above locations in time to help reveal the nature of the presented results.

Firstly, the motion past the rotor was investigated to understand azimuthally
where the temporal changes in the fluid motion were greatest. The temporal evolution
of the fluid motion past the rotor on the spanwise mid-plane ($z = 0.5$) for $D/R = 4$
yielded little change and did not give much information to link the fluid motion
characteristics with the aerodynamic performance discussed in section 6.4. Thus, no
results are given regarding fluid motion past the rotor on the spanwise mid-plane for
$D/R = 4$. The case where $D/R = 6$, however, displayed distinctly different velocity
fields for each position in time specified in figure 6.4. The velocity fields for $D/R = 6$
on the geometric mid-plane between the discs are shown in figure 6.5. Strong unsteady
activity was exhibited around the upper side of the rotor over the approximate range
$\pi/4 \leq \theta \leq 3\pi/4$. Position D notably saw the motion over the top of the cylinder act
approximately perpendicular to the uniform flow. Interestingly, position D in figure 6.5
corresponded to the highest generated lift of the specified positions in time in figure 6.4.
Recirculation was found in the trailing regions behind the cylinder for the majority time instants shown in figure 6.5. The circulation was commonly shown to develop at the disc edges where the tangential velocity is greatest. In the present study, the applied rotation rate $\alpha$ corresponds to the cylinder surface and, of course, the tangential velocity increases linearly with radial distance. Thus, the tangential velocity at the far edges of the discs will be greater than the velocity at the cylinder surface. In the present study, the tangential velocity is twice the cylinder tangential velocity for the case of D/R = 4 and three times the cylinder tangential velocity for the D/R = 6 case. Therefore, the assumption was made that the increased tangential velocity with increased disc diameter has a significant effect the development of circulation in the flow. The greater disc end tangential velocity also affected the intensity of the eddies generated within the cavity normal to the cylinder and the turbulence kinetic energy within the disc cavity which are discussed later in the chapter.

Depending on the position in time, circulation was found in other areas, sometimes rather interesting areas, around the rotor. One example was found at position D, the perpendicular motion mentioned earlier seemed to aid the creation of circulation slightly upstream of the rotor (i.e. $\theta = \pi/4$). The unsteady behaviour in the motion past the rotor is seemingly random or at least dependent on a variety of factors not entirely investigated by this study.

6.3.2 Fluid Motion on Cylinder-Normal Planes

As mentioned earlier, the lift is generated by a difference in pressure between the upper and lower sides of the rotor. Thus attention is next turned to flow behaviour in the plane of the cylinder axis for $\theta = \pi/2$ and $3\pi/2$ per the geometry defined by figure 6.3. The velocity fields on the upper side of the rotor are shown in figures 6.6 and 6.7 for D/R = 4 and D/R = 6 respectively. The flow was very complex in this region for discs examined. Radial outward motion was found in the near-disc wall regions in the upper and lower cavities due to the centrifugal nature of rotating discs creating near-wall Ekman layers. The upper side ($\theta = \pi/2$) consistently exhibited recirculation eddies form within the cavity. The development of these eddies was caused by the formation of the Ekman layer which grows with increasing radial distance. The interaction of this layer with the uniform exterior flow helps the eddies grow to large size where typically one eddy dominates the cavity. The possibility arises that these eddies grew too large within
the cavity thus inducing or contributing to the induced unsteadiness in the lift coefficient discussed later in this chapter. The cylinder normal planes in figures 6.6 and 6.7 show the temporally random positioning of the eddies between both discs. A closer examination of figure 6.6 indicates that, mainly at temporal positions A and C, that secondary eddies formed within the inner portion of the upper cavity. The exact pattern of motion of the recirculating eddies cannot be determined by the present study as it is believed to be random in nature, but further research on the recirculating behaviour within Thom disc cavities is needed.

The recirculating eddies found with discs with D/R = 6 (figure 6.7) were larger than those exhibited by discs with D/R = 4 (figure 6.6). The size of the eddies formed is presumably due to the difference in tangential velocity of the disc ends mentioned earlier. The presence of these large eddies raises the question of their effect on other parts of flow around the cylinder.

The corresponding streamwise motion on the axial planes is shown in figures 6.8 and 6.9. The streamwise motion is plotted on planes normal to the cylinder at $\theta = 0$ and $\pi$ per the geometry defined by figure 6.3. The upstream side ($\theta = 0$) of the rotor exhibited greater time-dependent motion than the downstream side since the motion within the cavity on the downstream side is essentially moving with the uniform flow. No relationship was found between the circulating flow over the upper side of the rotor, discussed earlier, and the upstream streamwise motion. The computation for D/R = 4 exhibited little temporal variation in the upstream velocity field ($\theta = 0$) shown in figure 6.8 when compared to that of the larger discs shown in figure 6.9. The larger disc simulation revealed a small amount of circulation in the cavity on the upstream side at select positions in time shown in figure (A, C, and E). Plane D in figure 6.9 corresponds to the position in time which generated the most lift. Plane D also showed a greater amount of circumferential motion on the upstream side indicating the flow was freely moving with the uniform flow. These results suggest higher lifts can be achieved when the unsteadiness in the flow dampened.

### 6.3.3 Unsteadiness within the Thom Disc Cavity

The circulating motion found in the previous figures indicated that a considerable amount of unsteadiness was exhibited by the presence of Thom discs. Planes of the turbulence kinetic energy are shown figures 6.10 and 6.11 for D/R = 4 and D/R = 6.
respectively. The planes shown are normal to the cylinder at $\theta = \pi/2$ and $3\pi/2$ per the geometry defined by figure 6.3. A comparison of the two figures exhibited similar kinetic energy structures on the upper side of the cylinder where the greatest variation in fluid motion was found. The $D/R = 6$ case showed greater levels of turbulence than the $D/R = 4$ case. Interestingly, the lower side ($\theta = 3\pi/2$) of the cylinder showed the greatest difference in kinetic energy planes between the two disc diameters. The lower side showed distinct concentrated areas of kinetic energy in the near-disc wall regions of each disc for the $D/R = 4$. The $D/R = 6$ case, however, found the kinetic energy was more diffuse throughout the cavity with the majority found in the near-wall regions (cylinder and discs) of the cavity. As mentioned earlier, the discrepancies in behaviour of the kinetic energy within the Thom disc cavity could be a result of the difference in tangential velocity.

Streamwise planes, using the geometry of streamwise planes defined earlier, of the kinetic energy are shown in figures 6.12 and 6.13 for $D/R = 4$ and $D/R = 6$ respectively. A similar increase in the magnitude of kinetic energy, as noted earlier, was observed for the increased disc diameter. The upstream side saw similar structures of kinetic energy to those found on the lower side of rotor discussed earlier (i.e. figures 6.10 and 6.11). As noted earlier the higher velocities associated with the edge region of the rotating disc for $D/R = 6$ seem the plausible case for the different flow structures observed. On the downstream side of the cylinder, the turbulence kinetic energy in figures 6.12 and 6.13 showed little difference for the two discs and little temporal change in the two cases as the centrifugal rotor flow is moving in the same direction as the uniform external flow.

6.4 Thom Disc Affected Aerodynamic Properties

The performance for Flettner rotors with Thom discs has been previously investigated by Thom (1934) and Craft et al. (2010, 2011). Thom (1934) carried out experiments for low-Re turbulent flows for various cavity dimensions while the latter reported CFD studies for one cavity geometry for $Re = 800,000$ with a range of $\alpha$. Craft et al. used a Thom disc cavity of dimensions $Z_{\text{disc}}/R = 2$ and $D/R = 4$ and found the addition of the Thom discs provided improved stability for $\alpha \geq 3$ but saw no significant improvement in the lift or drag. The present study used this information to investigate the effect of
Chapter 6: Flettner Rotor Flow with Thom Discs

Thom discs on a Flettner rotor for two new cavity geometries considered in this study. The lift and drag were computed for both cases to determine if aerodynamic performance improvements were found for Flettner rotors with larger, more axially compact Thom discs.

6.4.1 Rotor Lift Effects

As noted earlier from the stability analysis, the temporal evolution of the lift and drag coefficients revealed a decline in aerodynamic performance compared to the rotor geometry investigated in Craft et al. (2011). The first simulation investigated took the cavity geometry from Craft’s study and reduced the spanwise length by a factor of two, so the new cavity geometry essentially became a square cross-section. Though the case did not see the improved steadiness reported by Craft, although, it eventually settled to a steady oscillation around the same lift coefficient resolved by Craft. A steady state seemed to have been reached for approximately $t > 180$, shown in figure 6.14, as two fully repeated oscillations were recorded with no evidence the rotor flow would deviate from this pattern.

The next case extended the discs diameter to $D/R = 6$; a similar geometry to many of the flows investigated by Thom (1934). The results for this case saw a large reduction in the stability of the lift coefficient and consequently in the aerodynamic performance. Figure 6.14 revealed the lift force experienced large fluctuations up to approximately $\pm 50\%$ of the mean value. The fluctuations seem to be random with no tendency to approach an asymptotic value. As reported earlier, the behaviour of the fluid motion over the upper side of the rotor seemed to be random and unsteady. The unsteadiness in this region past rotor could link to the unsteady lift evolution exhibited for $D/R = 6$ in figure 6.14. Though both solutions converged computationally at each timestep, the temporal behaviour of the drag coefficient exhibited random unsteadiness for both discs. The reason for the aforementioned results remains uncertain and requires further research by possibly conducting a spectral analysis of the signal which could provide useful insight. Like the lift coefficient, larger fluctuations in the drag were found for the larger discs (figure 6.14).
6.4.2 Rotor Drag Effects

The temporal evolution of the drag coefficient exhibited small regions in time where the drag was negative for D/R = 4, while the drag coefficient was consistently negative for D/R = 6. The mean drag coefficients for the two cases in table 6.2 support the behaviour. Note, the drag coefficient given in table 6.2 for Craft et al. (2011) was for a Re = 800,000 making the difference between it and the present study coefficients justifiable. Though negative drag is not physical, this behaviour has been previously reported in some cases by Thom (1934). While on first consideration negative drag appears impossible, no violation of principles is involved. An explanation for the behaviour is obtained by considering the mean pressure contours around the rotor. Both cases are shown in figure 6.15 and a clockwise azimuthal shift in the region of low pressure on the lower side of the rotor was exhibited for the case with the larger discs. The shift in the low pressure region is further confirmed by the cylinder near-wall pressure distribution around the cylinder (figure 6.16) where the shift in this region was found to be approximately 6 degrees. Furthermore, since the drag coefficient in this study was (as usual) defined in the terms of the force acting in the opposite direction to the wind velocity, taking into consideration the clockwise shift in the pressure field, the corresponding negative value from table 6.2 was understandable.

Table 6.2: Mean lift and drag coefficients for present study and Craft et al. (2011).

<table>
<thead>
<tr>
<th>dim.</th>
<th>span</th>
<th>D</th>
<th>α</th>
<th>total Cl</th>
<th>total Cd</th>
</tr>
</thead>
<tbody>
<tr>
<td>3D</td>
<td>1R</td>
<td>4R</td>
<td>5</td>
<td>-12.0067</td>
<td>-0.0237</td>
</tr>
<tr>
<td>3D</td>
<td>1R</td>
<td>6R</td>
<td>5</td>
<td>-10.6557</td>
<td>-1.2730</td>
</tr>
<tr>
<td>Craft (2010)</td>
<td>2R</td>
<td>4R</td>
<td>5</td>
<td>≈-11.9798</td>
<td>≈0.25</td>
</tr>
</tbody>
</table>

6.4.3 Magnus Lift Comparison

The mean lift and drag coefficients were calculated and are presented in table 6.2. The case of shorter discs found a higher mean lift coefficient than the less stable case with larger discs. Despite the unsteadiness in the lift coefficient, the mean value for the smaller discs was very comparable to Craft et al. (2011). The smaller mean lift coefficient for D/R = 6 supported the poor overall performance found and discussed above for such large discs. Turning attention again to figure 6.16 and to the shift in the low pressure region for D/R = 6 discussed earlier, the shift implied the resultant surface...
force no longer acted perpendicular to the flow. In the present study, the Magnus lift was essentially computed by finding difference in pressure between the regions $0 \leq \theta \leq \pi$ and $\pi \leq \theta \leq 2\pi$. If the pressure distribution around the cylinder undergoes an azimuthal shift, as has been found, the computation of the “lift” might wish to take that into account since it is really the net thrust that is important.
Chapter 6: Flettner Rotor Flow with Thom Discs

6.5 Figures

Figure 6.1: Near-wall mesh resolution for (left) the 4R-TD mesh and (right) 6R-TD mesh.

Figure 6.2: 3D mesh resolution for rotor with Thom discs for (left) 4R-TD and (right) 6R-TD meshes.
Chapter 6: Flettner Rotor Flow with Thom Discs

Figure 6.3: Representation of the geometry and orientation for the simulated cavity found between a single set of Thom discs.

Figure 6.4: A particular temporal region of unsteadiness in the lift coefficient from $t = 150$ to $t = 175$. Labels indicate positions in time which will correspond to successive figures in this chapter.
Figure 6.5: Velocity field past the rotor for D/R = 6 at the geometric mid-plane between the discs z/R = 0.5 for temporal positions defined by figure 6.4. The outer ring indicates the outer edge of the disc.
Figure 6.6: Velocity field for $D/R = 4$ on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond for figure 6.4.
Chapter 6: Flettner Rotor Flow with Thom Discs

Figure 6.7: Velocity field for D/R = 6 on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond for figure 6.4.
Chapter 6: Flettner Rotor Flow with Thom Discs

Figure 6.8: Velocity field for D/R = 4 on cylinder normal planes at $\theta = 0$ and $\pi$. Labels correspond for figure 6.4.
Figure 6.9: Velocity field for D/R = 6 on cylinder normal planes at θ = 0 and π. Labels correspond for figure 6.4.
Figure 6.10: Plotted turbulence energy for D/R = 4 on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond with figure 6.4; range given is for mean k data.
Figure 6.11: Plotted turbulence energy for D/R = 6 on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond with figure 6.4; range given is for mean k data.
Figure 6.12: Plotted turbulence energy for D/R = 4 on cylinder normal planes at $\theta = 0$ and $\pi$. Labels correspond with figure 6.4; range given is for mean k data.
Figure 6.13: Plotted turbulence energy for D/R = 6 on cylinder normal planes at \( \theta = 0 \) and \( \pi \). Labels correspond with figure 6.4; range given is for mean k data.

Figure 6.14: Temporal evolution of the lift (solid lines) and drag (dashed lines) coefficients both present simulations with Thom discs for Re = 140k, also includes results from Craft et al (2011) for Re = 800k; all cases with \( \alpha = 5 \).
Chapter 6: Flettner Rotor Flow with Thom Discs

Figure 6.15: Mean pressure contours at mid-span (z = 0.5) for (left) D/R = 4 and (right) D/R = 6.

Figure 6.16: Mean cylinder wall pressure distribution. θ follows the orientation in figure 6.3, reference pressure chosen for Cp = 1 at θ = 0 for α = 0.
7 Concluding Remarks and Future Work

The present three-dimensional, time-dependent, numerical study of Flettner rotor flows has provided new information regarding the effect of the axial extent of the solution domain smooth cylinder and the modification to the flow that result from the addition of circumferential discs applied to the rotor. These are summarized in the first two sections and section 7.3 discusses suggested further numerical research to be taken to further improve the performance of the Flettner rotor.

7.1 Effects of Span Length on Solution Domain

3D simulations of flow over a cylinder of infinite length were thought to be independent of the length of cylinder section applied in the solution domain per the work by Breuer (2000) for a non-rotating cylinder. However, instantaneously inducing translation and rotation on a smooth cylinder has been known to have great effect on the flow. The present study has therefore explored the possible dependency of the numerical results on the solution domain’s cylinder span length needed to be examined further.

The present brief CFD study examined 3D rotating smooth cylinder flows for $Re = 140,000$ for two different rotation rates $\alpha = 2$ and 5. The former was examined in comparison with the only turbulent large-eddy simulation (Karabelas, 2010) currently available, and the latter was examined as it was consistent with spin rates at which an actual Flettner-rotor driven vessel may expect to operate. The Reynolds number was chosen to match that of Karabelas (2010) though it is not representative of wind flows expected to act on an actual rotor driven vessel.

For each rotational speed, two cylinder span lengths were examined within the solution domain. The 3D solutions were also compared with the author’s 2D computations. The main findings were summarized as follows:
A) $\alpha = 2$

- The flow simulations exhibited little or no discrepancies in the fluid motion about the cylinder via the mean streaklines for the span lengths examined. The spanwise motion along the cylinder behaved likewise indicating the flow for dimensionless spin ratios up to 2 was essentially two-dimensional.
- The above eddy structures found via the streaklines were non-shedding though the cylinder lift coefficient showed very small oscillations suggesting mild periodic fluctuations in the near-wake structures.
- Increased span length had little effect on the pressure distribution around the cylinder in the near-wall region and thus the aerodynamic properties showed effectively no differences concluding the aerodynamic performance of the Flettner rotor remained unaffected for relatively low spin rate.

B) $\alpha = 5$

- Increased spin rate exhibited spanwise undulation in the near-wall region around the cylinder with defined repeated structures found along the cylinder. The longer span showed these structures were greater in clarity and greater in number than the shorter span indicating an inability of the shorter span to adequately capture the fluid flow.
- Further investigation determined that the spanwise undulation developed streamwise circulation around the cylinder, the greatest of which was found over the upper side. The circulation yielded trailing vortices commonly found with both span length domains.
- However, the shorter span exhibited downstream interaction between the trailing vortices which was not present in the case of the longer span. The results indicated the periodic boundary condition applied at the top and bottom ends of the cylinder was responsible for the seemingly unphysical downstream motion.
- The increase in dimensionless span length of the solution domain from 2 to 6 led to a small difference in the pressure distribution around the cylinder and thus a small difference in the aerodynamic performance was notable.
- The longer solution domain showed a small increase in lift over the shorter domain though the change may not be significant given other possible sources of
error (for example in the turbulence model) to compensate for the additional computational effort of the larger domain.

## 7.2 Effect of Thom Discs on the Flettner Rotor

Previous URANS studies of Craft et al. (2010, 2011) concluded the addition of Thom discs on a cylinder at low rotation speeds dampened fluctuations in the lift coefficient but recorded no significant effect on the magnitude of the lift. Higher spin rates (i.e. $\alpha > 3$) exhibited a small increase in the lift but this was still less than that found in the experimental investigation by Thom (1934). Furthermore, the discs examined by Thom were larger in diameter and spaced closer than Craft’s numerical investigation. Therefore, it appeared that with larger disc placed more compactly along the rotor, improved lift coefficients were possible. This proposition has been numerically examined in the present study.

Two cases were examined for $Re = 140,000$ which was recognizably much lower than an actual Flettner-rotor vessel would experience, but suitable for the main aim of the present study. The disc spacing for both cases was half that examined by Craft et al. (2010, 2011). Disc diameters of 4R and 6R were examined based on the results of Thom (1934).

**A) Effects of Thom Discs on Fluid Motion**

- The smaller discs exhibited no significant temporal changes in the fluid motion past the rotor; however the larger discs, with diameter increased to 6R, found substantial temporal evolution of the fluid motion primarily on the upper side of the rotor. Circulation was commonly found in the near-wake region at the disc ends where the local tangential velocity is at a maximum.

- Fluid motion within the Thom disc cavity over the upper side of the cylinder saw recirculating eddies form for both disc diameters examined. Typically a single eddy would dominate the cavity and the position of the said dominant eddy seemed to be temporally random in time. The larger disc case saw larger eddies form and was proposed that the increased tangential velocity was responsible.
Chapter 7: Concluding Remarks and Future Work

- Attention was given to the turbulence kinetic energy within the cavity where the larger discs exhibited greater distortion of the energy within the cavity compared to the energy exhibited by the shorter disc case. Again, the idea regarding increased tangential velocity with larger discs was deemed responsible for the difference in behaviour.

B) Effects of Thom Discs on Aerodynamic Performance

- Decreased stability in the lift coefficient was exhibited by both present cases compared to that examined by Craft et al. (2011). The increased disc diameter to 6R exhibited very large fluctuations in the lift coefficient, seemingly random in nature.
- Furthermore, the smaller disc case found mean higher lift coefficient than the larger disc case.
- Further analysis of the pressure around the rotor found, in the case of larger discs, the low pressure region on the lower side was no longer orthogonal to the streamwise direction and actually shifted azimuthally clockwise around the cylinder by approximately 6 degrees. The shift arguably made a contribution to the lower mean lift coefficient found with larger discs.
- The reported mean drag coefficients were found to be negative and thus apparently unphysical. However, similar results have been previously reported by Thom (1934) for Flettner rotor flows with Thom discs. The drag was increasingly negative for the larger discs where an azimuthal shift in the low pressure region provided a possible explanation for the consistently negative (in time) drag coefficient. The low pressure region shifted clockwise beyond $\theta = 0$ which is where the drag is calculated in the present study.

7.3 Future Work

The present study has hopefully advanced our understanding of the performance of the Flettner-rotor driven sea vessels. Nonetheless, a great amount of further investigation is needed to provide more details regarding the following issues. For the case of a smooth cylinder flows, an examination of higher Re flows well within the probably range to be experienced by an actual Flettner-rotor craft is urgent. Examining the flow with a more
advanced turbulence model, such as a non-linear eddy-viscosity turbulence model, would deliver more realistic simulations than the linear EVM in the present study. A more complete study for a range of higher spin rates in high Re to understand the reported asymptotic limit in lift previously reported for winds speeds representative of those expected in actual Flettner rotor use. Lastly, it would be very informative (though numerically challenging) to incorporate an entire rotor within the solution domain to be studied, including end effects (primarily those of a vessel’s deck simply modelled by a plane) and the effect of varied wind speed along the length of the cylinder.

The proposal of adding discs to gain improvements in rotor aerodynamics must be investigated further. There is a desire to upgrade the near-wall treatment presently covered by wall functions after a study by Zacharos (2010) showed that the flow near a rotating disc surface is largely different if the wall function is modified to allow velocity skewing. With this method, he was able to reproduce the ‘Ekman spirals’ that form next to spinning discs. The information provided in the present study and that of Thom (1934) were found to be greatly different and further experimental investigation is needed. Also, a study incorporating further untested geometries with Thom discs would provide a better understanding of the flow behaviour with changes in geometry. A numerical study using LES such as provided by Karabelas (2010) for rotating smooth cylinder flows would be beneficial in a similar study with the addition of Thom discs. Furthermore, in a comparison effort partnered with the potential smooth cylinder investigation, a study of an entire rotor with Thom discs attached will eventually need to be pursued to understand the effects between adjacent disc cavities as Craft et al. (2011) mentioned that neither a periodic or symmetry boundary conditions applied on the rotor ends was correct. This last development may be in the distant future of Flettner rotor research.
References


References


References


Thom, A. 1934. Effects of discs on the air forces on a rotating cylinder. ARC R&M 1623.


Figure A.7.1: Plotted $P$ for $D/R = 4$ on cylinder normal planes at $\theta = \pi/2$ and $3\pi/2$. Labels correspond with figure 6.4.
Appendices

Figure A.7.2: Plotted $P$ for $D/R = 6$ on cylinder normal planes at $0 = \pi/2$ and $3\pi/2$. Labels correspond with figure 6.4.
Figure A.7.3: Plotted $P$ for $D/R = 4$ on cylinder normal planes at $0 = 0$ and $\pi$. Labels correspond with figure 6.4; range given is for mean $P$ data.
Appendices

Figure A.7.4: Plotted P for D/R = 6 on cylinder normal planes at $\theta = 0$ and $\pi$. Labels correspond with figure 6.4; range given is for mean P data.
Figure A.7.5: Spanwise motion (W/R) around the rotor for D/R = 4 at z/R = 0.5 (per geometry in figure 6.3) for temporal positions defined by figure 6.4. The outer ring indicates the outer edge of the disc.
Figure A.7.6: Spanwise motion (W/R) around the rotor for D/R = 6 at z/R = 0.5 (per geometry in figure 6.3) for temporal positions defined by figure 6.4. The outer ring indicates the outer edge of the disc.