INVESTIGATION OF FLOW PHYSICS OF PUMP INTAKE FLOWS USING LARGE EDDY SIMULATION

by

Talia Ekin Tokyay and S. George Constantinescu

IIHR Technical Report No. 445

IIHR—Hydroscience & Engineering
College of Engineering
The University of Iowa
Iowa City, Iowa 52242-1585 USA

July 2005
ACKNOWLEDGMENTS

I would like to thank Wu Yulin, Li Yong and Li Xiaoming, members of the Thermal Engineering Department of Tsinghua University, Beijing, China, for providing their experimental data to validate our pump intake model. The pump intake geometry was provided by Hitachi Industries, Tsuchiura-shi, Japan.

This study could not be conducted without the special help and enthusiasm of my advisor Dr. George Constantinescu. He was very helpful and motivating in all steps of our research. I would like to thank IIHR-Hydroscience and Engineering which supported this study by its internal funds. I would like to thank Dr. Virendra C. Patel and Dr. Robert Ettema, for their review and invaluable recommendations on this report.
ABSTRACT

Large Eddy Simulation (LES) is used to simulate the flow in a pressurized pump intake of realistic geometry and to investigate the dynamics of the coherent structures including phenomena such as intermittency and meandering that characterize the dynamics of the vortices present in pump bays. The present study presents, for the first time, results of a numerical investigation of pump intake flows using a dynamic LES turbulence model without wall functions, a non-dissipative viscous solver and fine enough computational grids that can resolve the energetically important eddies in the flow for the given flow conditions. The other novelty of the present study is that phenomena such as vortex intermittency and meandering are captured as a part of the time developing LES solution. To validate the model results of the LES simulation are compared with the data available from the Particle Image Velocimetry (PIV) experiments conducted for a pump intake of identical geometry by Yulin et al. (2000) and to a RANS simulation. It is shown that even if both models can fairly successfully capture the mean velocity distribution and flow features around the pump column, LES can more accurately predict the mean flow and turbulence statistics compared to the steady RANS model, especially the distribution of the mean turbulent kinetic energy (t.k.e.) and mean fluctuations. In the second part of the study, the LES flow fields are used to investigate the highly unsteady dynamics of the main coherent vortices in the flow including their spectral content. Present LES results confirm the presence of highly unstable wall attached vortices that move within the bay and change considerably their strength, core size and structure as they interact with other main vortices or with the surrounding turbulence. This study shows that LES can be used to obtain detailed information on the evolution and interactions among the main vortices that is very hard to obtain from scaled model studies, even if advanced techniques such as Particle Image Velocimetry (PIV) techniques are employed. This is because the information obtained at a certain time
using PIV can cover only one 2D plane. As the vortices are typically unsteady, meandering and/or intermittent this partial description of the flow may not be enough to fully understand the overall vortical structure inside the pump bay area. LES does not face these limitations, as the whole 3D fields are available during the simulated time interval. The long time goal of the present work is to use LES as a predictive tool that can be employed in the design or redesign process of pump intakes.
<table>
<thead>
<tr>
<th>TABLE OF CONTENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF FIGURES ........................................................................................................... vi</td>
</tr>
<tr>
<td>CHAPTER 1. INTRODUCTION ........................................................................................ 1</td>
</tr>
<tr>
<td>1.1 Overview ................................................................................................................. 1</td>
</tr>
<tr>
<td>1.2 Background ............................................................................................................... 2</td>
</tr>
<tr>
<td>1.3 Scope and Objectives of the Present Study ............................................................. 4</td>
</tr>
<tr>
<td>1.4 Review of Literature ............................................................................................... 4</td>
</tr>
<tr>
<td>1.4.1 Review of Relevant Experimental Studies ........................................................... 4</td>
</tr>
<tr>
<td>1.4.1.1 Previous Experimental Studies ........................................................................ 5</td>
</tr>
<tr>
<td>1.4.1.1.1 Free Surface Vortices ................................................................................. 5</td>
</tr>
<tr>
<td>1.4.1.1.2 Subsurface Vortices ..................................................................................... 7</td>
</tr>
<tr>
<td>1.4.1.1.3 Turbulent Swirling Pipe Flow ..................................................................... 7</td>
</tr>
<tr>
<td>1.4.1.2 The Experimental Study by Ansar (1997) ......................................................... 8</td>
</tr>
<tr>
<td>1.4.1.3 The Experimental Study by Rajendran (1998) ............................................... 9</td>
</tr>
<tr>
<td>1.4.1.4 The Experimental Study by Yulin et al. (2000) ............................................. 11</td>
</tr>
<tr>
<td>1.4.2 Review of the Numerical Studies ........................................................................ 13</td>
</tr>
<tr>
<td>1.4.2.1 Previous Numerical Studies ............................................................................ 13</td>
</tr>
<tr>
<td>1.4.2.2 The Numerical Study by Ansar (1997) ......................................................... 14</td>
</tr>
<tr>
<td>1.4.2.3 The Numerical Study by Constantinescu (1998) ......................................... 15</td>
</tr>
<tr>
<td>1.4.2.4 The Numerical Study by Li et al. (2001) ...................................................... 16</td>
</tr>
<tr>
<td>CHAPTER 2. NUMERICAL METHODS ......................................................................... 46</td>
</tr>
<tr>
<td>2.1 Introduction .............................................................................................................. 46</td>
</tr>
<tr>
<td>2.2 Reynolds Averaged Navier-Stokes Model .............................................................. 46</td>
</tr>
<tr>
<td>2.2.1 General Description of SST Model ..................................................................... 47</td>
</tr>
<tr>
<td>2.2.2 Transport Equations and Parameters for the SST Model Implemented in Fluent . 47</td>
</tr>
<tr>
<td>2.3 Large Eddy Simulation Model .............................................................................. 50</td>
</tr>
<tr>
<td>CHAPTER 3. DESCRIPTION OF COMPUTATIONAL DOMAIN, MESH, FLOW AND BOUNDARY CONDITIONS ................................................................. 54</td>
</tr>
<tr>
<td>3.1 Introduction .............................................................................................................. 54</td>
</tr>
<tr>
<td>3.2 Physical Description of the Domain and Test Conditions .................................... 54</td>
</tr>
<tr>
<td>3.3 Boundary Conditions ............................................................................................. 55</td>
</tr>
<tr>
<td>3.3.1 Inlet Sections .................................................................................................... 55</td>
</tr>
<tr>
<td>3.3.2 Outlet ............................................................................................................... 56</td>
</tr>
<tr>
<td>3.3.3 Walls ............................................................................................................... 56</td>
</tr>
<tr>
<td>3.4 Grid Generation ..................................................................................................... 56</td>
</tr>
<tr>
<td>3.5 Discussions of Convergence and Computational Resources ................................ 58</td>
</tr>
<tr>
<td>CHAPTER 4. DISCUSSION OF RESULTS ................................................................ 63</td>
</tr>
<tr>
<td>4.1 Validation: Comparison of Results from Experiment, RANS and LES ............... 63</td>
</tr>
<tr>
<td>4.2 3D Visualizations of the Main Vortical Structures ............................................ 68</td>
</tr>
<tr>
<td>4.3 Analysis of Instantaneous Flow Fields .................................................................. 70</td>
</tr>
<tr>
<td>4.4 Time Series and Power Spectra of Instantaneous Quantities .............................. 74</td>
</tr>
<tr>
<td>4.5 Decay of Vorticity inside the Pump Column ....................................................... 75</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

Figure 1.1 General view of a simplified pump intake configuration used in the study by Constantinescu and Patel (1998).................................................................19

Figure 1.2 General view of the pump intake geometry described in the study by Ansar (1997) .............................................................................................................20

Figure 1.3 Experimental setup of Ansar et al. (1997) for approach flow with no-cross flow case ..........................................................................................................21

Figure 1.4 Experimental setup of Ansar et al. (1997) for approach flow with cross flow case .............................................................................................................22

Figure 1.5 Sketch of the observed approach flow distribution for the no-cross flow from the work of Ansar et al. (1997) ........................................................................23

Figure 1.6 Sketch of the observed approach flow distributions for the cross flow from the work of Ansar (1997) .................................................................................23

Figure 1.7 Axial velocity distribution at the pump throat for the no cross flow case in the study by Ansar (1997).....................................................................................24

Figure 1.8 Axial velocity distribution at the pump throat for the cross flow case in the study by Ansar (1997).........................................................................................24

Figure 1.9 Experimental setup of the pump bay used in the study by Rajendran (1998)........................................................................................................................25

Figure 1.10 Sketch of vortices observed in the model pump intake in study by Rajendran (1998) .............................................................................................................25

Figure 1.11 Variation of instantaneous strength of each vortex as a function of time measured by Rajendran (1998).................................................................................26

Figure 1.12 Visualization of the floor attached vortex using LIF in the experiments of Rajendran (1998). Picture is taken from the channel bottom ......................26

Figure 1.13 Visualization of side wall 1 vortex by LIF in the experiments of Rajendran (1998) .............................................................................................................27

Figure 1.14 Instantaneous velocity vectors and out-of-plane vorticity contours in a plane parallel to the bottom cutting through the floor attached vortex from the experimental study by Rajendran (1998) .............................................................................27

Figure 1.15 Average velocity vectors and out-of-plane vorticity contours in a plane parallel to the channel bottom cutting through the floor attached vortex from the experimental study by Rajendran (1998) .................................................................28

Figure 1.16 Meandering of floor attached vortex measured by Rajendran (1998) [Pipe column projection on the floor is shown with dashed line] ............................28
Figure 1.17 Average velocity vectors and out-of-plane mean vorticity contours in a plane situated near the free surface showing the free surface vortices from the experimental study by Rajendran (1998)..............................29

Figure 1.18 Meandering of the free surface vortex measured by Rajendran (1998)........29

Figure 1.19 General view of the pump intake geometry considered in the experiments of Yulin et al. (2000).................................................................30

Figure 1.20 Detailed schematic of the pump bell of the pump intake geometry considered in the experiments of Yulin et al. (2000).........................................................31

Figure 1.21 Computational grid for the single pump intake simulated in the numerical study by Ansar (1997)..........................................................32

Figure 1.22 Computational grid for the pump intake geometry containing two pump columns simulated by Ansar (1997)......................................................33

Figure 1.23 Streamlines patterns calculated by Ansar (1997) in the pump intake containing two pump columns; a) Streamlines at the free surface in the case of equal discharges in the pumps (Q1 = Q2); b) Streamlines in plane situated between floor and pump bell level showing the floor attached vortices in the case of equal discharges in the pumps (Q1 = Q2); c) Streamlines at the free surface in the case of unevenly split discharges in the pumps (Q1>Q2); d) Streamlines in plane situated between floor and pump bell level showing the floor attached vortices in the case of unevenly split discharges in the pumps (Q1>Q2)..........................................................34

Figure 1.24 Intake configuration used in the validation study by Constantinescu (1998)........................................................................................................35

Figure 1.25 Streamlines at the free surface from the study by Constantinescu (1998).....36

Figure 1.26 Visualization of free surface vortices using vorticity contours from the study by Constantinescu (1998).................................................................36

Figure 1.27 Comparison of the floor attached vortex structure (streamlines and out of plane vorticity from experiments and simulations; a) general flow pattern at a plane situated between the floor and the pump column entry level; b) numerical simulations of Constantinescu (1998); c) experimental results of Rajendran (1998) .................................................................37

Figure 1.28 Comparison of the side wall attached vortex structure (streamlines and out of plane vorticity from experiments and simulations; a) general flow pattern at a plane situated between side wall 2 and the pump column; b) numerical simulations of Constantinescu (1998); c) experimental results of Rajendran (1998) ........................................................................................................38

Figure 1.29 Visualization of; a) main free surface vortex; b) corner vortex using particle path lines from the simulations of Constantinescu (1998) .......................39

Figure 1.30 Comparison between the streamlines of the floor attached vortex obtained from the numerical simulations of Li et al. (2001) and Constantinescu (1998) and the experimental results of Rajendran (1998)........40
Figure 1.31 Location of the velocity measurement stations in the approach channel in the study by Li et al. (2001). The pump column contains a pump bell similar to the experiment of Ansar (1997).................................................................41

Figure 1.32 Comparison of the streamwise velocity profiles obtained from the simulations of Li et al. (2001) and the experiments of Ansar (1997) for the case of no cross flow at sections B, C and D (see Figure 1.31).................................42

Figure 1.33 Comparison of the streamwise velocity profiles obtained from the simulations of Li et al. (2001) and the experiments of Ansar (1997) for the case with cross flow at sections B, C and D (see Figure 1.31).................................43

Figure 1.34 Configuration of the pump intake geometry, including the far field, simulated in the study by Li et al. (2001).................................................................................................44

Figure 1.35 Configuration of the pump intake geometry (near field) simulated in the study by Li et al. (2001)......................................................................................................................44

Figure 1.36 Qualitative comparison between the numerical simulations and the experiments from the study by Li et al. (2001) in a longitudinal plane cutting through the symmetry axis of the pump column. .................................................................................45

Figure 3.1 General view of the pump intake geometry considered in the present study................................................................................................................................................59

Figure 3.2 Grid used in Reynolds averaged Navier-Stoke simulations........................................60

Figure 3.3 Grid used in Large Eddy Simulations ........................................................................60

Figure 3.4 Detailed view of the grid of the top wall of the geometry used in LES.......................60

Figure 3.5 LES grid of the present study a) view of the grid at a section through the center of the pipe; b) detail of the grid near the wall of the pump column.............................61

Figure 3.6 View of the grid at a section parallel to the channel bottom cutting through the pump column at a distance of 1.425D from the floor of the pump intake.................................................................................62

Figure 4.1 Sections where experimental measurements, RANS and LES results are compared................................................................................................................................................78

Figure 4.2 2D streamlines from experiment, RANS and LES in x2 plane situated at 0.92D away from side wall 1........................................................................................................79

Figure 4.3 2D streamlines from experiment, RANS and LES in x3 plane situated at 1.21D away from side wall 1.................................................................................................79

Figure 4.4 2D streamlines from experiment, RANS and LES in x4 plane cutting through the center of the pump column........................................................................................................80

Figure 4.5 2D streamlines from experiment, RANS and LES in x5 plane situated at 1.21D away from side wall 2.................................................................................................80
Figure 4.6 2D streamlines from experiment, RANS and LES in x7 plane situated at 0.77D away from side wall 2 ................................................................. 81

Figure 4.7 2D streamlines from experiment, RANS and LES in z1 plane situated at 0.4D away from channel bottom .......................................................... 81

Figure 4.8 2D streamlines from experiment, RANS and LES in z3 plane situated at 0.9D away from channel bottom ......................................................... 82

Figure 4.9 2D streamlines from experiment, RANS and LES in z5 plane situated at 1.425D away from channel bottom ..................................................... 82

Figure 4.10 Turbulent kinetic energy contours in x4 plane cutting through the center of the pump column ................................................................. 83

Figure 4.11 Velocity magnitude contours in x6 plane situated at 0.92D away from side wall 2 ................................................................. 83

Figure 4.12 Velocity magnitude contours in x7 plane situated at 0.77D away from side wall 2 ................................................................. 84

Figure 4.13 Turbulent kinetic energy contours in x7 plane situated at 0.77D away from side wall 2 ................................................................. 84

Figure 4.14 Velocity magnitude contours y1 plane situated at 0.4D away from the back wall ................................................................. 85

Figure 4.15 Turbulent kinetic energy contours y1 plane situated at 0.4D away from the back wall ................................................................. 85

Figure 4.16 Velocity magnitude contours in y3 plane cutting through the center of the pump column ................................................................. 86

Figure 4.17 Turbulent kinetic energy contours in y3 plane cutting through the center of the pump column ................................................................. 86

Figure 4.18 Velocity magnitude contours in y4 plane situated at 0.77D away from the center of the pump column parallel to the back wall ................................................. 87

Figure 4.19 Velocity magnitude contours in z1 plane situated at 0.4D away from channel bottom ................................................................. 87

Figure 4.20 Turbulent kinetic energy contours in z1 plane situated at 0.4D away from channel bottom ................................................................. 87

Figure 4.21 Out-of-plane absolute vorticity contours in z1 plane situated at 0.4D away from channel bottom ................................................................. 88

Figure 4.22 Velocity magnitude contours in z2 plane situated at 0.6D away from channel bottom ................................................................. 89

Figure 4.23 Turbulent kinetic energy contours in z2 plane situated at 0.6D away from channel bottom ................................................................. 89
Figure 4.24 Out-of-plane absolute vorticity contours in z2 plane situated at 0.6D away from channel bottom .................................................................90

Figure 4.25 Out-of-plane vorticity contours in z5 plane situated at 1.425D away from channel bottom ....................................................................................90

Figure 4.26 Visualization of floor attached vortex; using 3D streamlines from mean LES solution (statistics) .............................................................................91

Figure 4.27 Visualization of floor attached vortex using 3D streamlines from an instantaneous LES solution..................................................................................91

Figure 4.28 Visualization of side wall attached vortex using 3D streamlines from mean LES solution (statistics) .................................................................92

Figure 4.29 Visualization of side wall attached vortex using 3D streamlines from an instantaneous LES solution........................................................................92

Figure 4.30 Visualization of the vertical vortex between pump column and side wall using 3D streamlines from mean LES solution (statistics) .........................93

Figure 4.31 Visualization of the vertical vortex between pump column and side wall using 3D streamlines from an instantaneous LES solution ..................93

Figure 4.32 Instantaneous streamlines in the symmetry plane x4 cutting through the center of the pipe and parallel to the side walls; a) t = 0; b) t = 15D/U0; c) t = 30D/U0; d) t = 45D/U0; e) t = 60D/U0; f) t = 75D/U0; g) t = 90D/U0; h) t = 105D/U0; i) t = 120D/U0; j) mean streamlines from averaged LES solution ...............95

Figure 4.33 Instantaneous streamlines in x6 plane parallel to the side wall situated at 0.92D from side wall 2; a) t = 0; b) t = 15D/U0; c) t = 30D/U0; d) t = 45D/U0; e) t = 60D/U0; f) t = 75D/U0; g) t = 90D/U0; h) t = 105D/U0; i) t = 120D/U0; j) mean streamlines from averaged LES solution ..................................................96

Figure 4.34 Instantaneous streamlines in z2 plane parallel to channel floor at the pump bell level (0.6D from the floor); a) t = 0; b) t = 15D/U0; c) t = 30D/U0; d) t = 45D/U0; e) t = 60D/U0; f) t = 75D/U0; g) t = 90D/U0; h) t = 105D/U0; i) t = 120D/U0; j) mean streamlines from averaged LES solution ...............................97

Figure 4.35 Instantaneous streamlines in z4 plane parallel to channel bottom situated 1.13D from channel bottom; a) t = 0; b) t = 15D/U0; c) t = 30D/U0; d) t = 45D/U0; e) t = 60D/U0; f) t = 75D/U0; g) t = 90D/U0; h) t = 105D/U0; i) t = 120D/U0; j) mean streamlines from averaged LES solution .................................98
Figure 4.38 Instantaneous out-of-plane vorticity contours in the symmetry plane \( x_4 \) cutting through the center of the pipe and parallel to the side walls; a) \( t = 0 \); b) \( t = 15D/U_0 \); c) \( t = 30D/U_0 \); d) \( t = 45D/U_0 \); e) \( t = 60D/U_0 \); f) \( t = 75D/U_0 \); g) \( t = 90D/U_0 \); h) \( t = 105D/U_0 \); i) \( t = 120D/U_0 \); j) mean out-of-plane vorticity from averaged LES solution.

Figure 4.39 Instantaneous out-of-plane vorticity contours in \( x_6 \) plane parallel to the side wall situated at 0.92D from side wall 2; a) \( t = 0 \); b) \( t = 15D/U_0 \); c) \( t = 30D/U_0 \); d) \( t = 45D/U_0 \); e) \( t = 60D/U_0 \); f) \( t = 75D/U_0 \); g) \( t = 90D/U_0 \); h) \( t = 105D/U_0 \); i) \( t = 120D/U_0 \); j) mean out-of-plane vorticity from averaged LES solution.

Figure 4.40 Instantaneous out-of-plane vorticity contours in \( z_2 \) plane parallel to channel floor at the pump bell level (0.6D from the floor); a) \( t = 0 \); b) \( t = 15D/U_0 \); c) \( t = 30D/U_0 \); d) \( t = 45D/U_0 \); e) \( t = 60D/U_0 \); f) \( t = 75D/U_0 \); g) \( t = 90D/U_0 \); h) \( t = 105D/U_0 \); i) \( t = 120D/U_0 \); j) mean out-of-plane vorticity from averaged LES solution.

Figure 4.41 Instantaneous out-of-plane vorticity contours in \( z_4 \) plane parallel to the channel bottom situated 1.13D from channel floor; a) \( t = 0 \); b) \( t = 15D/U_0 \); c) \( t = 30D/U_0 \); d) \( t = 45D/U_0 \); e) \( t = 60D/U_0 \); f) \( t = 75D/U_0 \); g) \( t = 90D/U_0 \); h) \( t = 105D/U_0 \); i) \( t = 120D/U_0 \); j) mean out-of-plane vorticity from averaged LES solution.

Figure 4.42 Meandering of the core of the floor attached vortex in a section situated at 0.2D from the channel floor in time.

Figure 4.43 Evolution of the mean core radius of the floor attached vortex in a section situated at 0.2D from the channel floor.

Figure 4.44 Evolution of the absolute circulation of the floor attached vortex in a section situated at 0.2D from the channel floor.

Figure 4.45 Meandering of the core of the side wall 2 attached vortex in a section situated at 0.77D from the side wall 2.

Figure 4.46 Spectral analysis of floor attached vortex; a) pressure time series and power spectra at a point situated near the channel floor below the pump bell level; b) pressure time series and power spectra at a point situated at the pump bell level.

Figure 4.47 Spectral analysis of floor attached vortex at a point inside the pump column; a) y-velocity time series and power spectra; b) pressure time series and power spectra; c) x-velocity power spectrum in log-log scale and in linear-linear scale; d) z-velocity power spectrum in log-log scale and low and in linear-linear scale.

Figure 4.48 Spectral analysis of side wall 2 attached vortex; a) time series of the pressure; b) pressure power spectrum in log-log scale; c) low frequency part of pressure spectrum; d) high frequency part of pressure spectrum.
Figure 4.49 Spectral analysis at a point situated near the junction between the back wall and the upper wall; a) time series of the pressure; b) pressure power spectrum in log-log scale; c) low frequency part of pressure spectrum; d) high frequency part of pressure spectrum.

Figure 4.50 Mean out-of-plane vorticity contours at different levels inside the pipe obtained from the statistics of LES; a) level $z = 0$; b) level $z = 0.6D$; c) level $z = 1.0D$; d) level $z = 1.5D$; e) level $z = 1.7D$; f) level $z = 2.0D$; g) level $z = 2.5D$; h) level $z = 3.0D$.

Figure 4.51 Decay of absolute circulation inside the pump column.

Figure 4.52 Circulation inside the pump column in a plane 0.6D from the pump bell level; a) evolution of circulation in time; b) mean out-of-plane vorticity contours; c) instantaneous out-of-plane vorticity contours.

Figure 4.53 Circulation inside the pump column in a plane 2.0D from the pump bell level; a) evolution of circulation in time; b) mean out-of-plane vorticity contours; c) instantaneous out-of-plane vorticity contours.
CHAPTER 1. INTRODUCTION

1.1 Overview

In the present research, Large Eddy Simulation (LES) was used to simulate the flow in a pressurized pump intake of realistic geometry and to investigate the dynamics of the coherent structures. The present study presents, for the first time, results of a numerical investigation of pump intake flows using:

1) a dynamic LES turbulence model without wall functions
2) a non-dissipative viscous solver
3) fine enough computational grids needed to resolve the energetically important eddies in the flow for the given flow conditions.

The other novelty of the present study is that for the first time, phenomena such as vortex intermittency and meandering which are captured as a part of the numerical simulation. This is possible because we are using an eddy resolving technique. This capability of the model is relevant as most of the reticence of the industry to use Computational Fluid Dynamics (CFD) to address problems at pump intakes comes from the fact that these types of phenomena which are widely present within pump bays cannot be captured by the RANS based CFD codes used in consulting.

In the first part of the study, the results of LES are compared to results obtained using a Reynolds Averaged Navier-Stokes (RANS) model implemented in a state-of-the-art commercial CFD software, Fluent, to better put in perspective the performance of the LES model. Results of the simulations are compared against the data available from the Particle Image Velocimetry (PIV) experiments conducted for a pump intake of identical geometry by Yulin et al. (2000). It is shown that even if both models can fairly successfully capture the mean velocity distribution and flow features around the pump column, LES can more accurately predict the mean flow and turbulence statistics compared to the steady RANS model, especially the distribution of the mean turbulent kinetic energy (t.k.e.) and mean fluctuations.

In the second part of the study, the LES flow fields are used to investigate the highly unsteady dynamics of the main coherent vortices in the flow including their spectral content. LES allows a better insight into understanding the interaction between
these vortices which are most of the time intermittent, unstable and move around the bay in a chaotic fashion even compared with PIV investigations. This is because in LES the full 3D flow fields are available at each time step for analysis as opposed to PIV where the flow in only one selected 2D section can be investigated at a time. As the main vortices present in the flow studied in present investigation are not steady and the movements of their cores are not periodic or even quasi periodic, it is very difficult to understand the interaction between the main vortices based on recordings done over different time intervals. Present LES results confirm the presence of highly unstable wall attached vortices that move within the bay and change considerably their strength, core size and structure as they interact with other main vortices or with the surrounding turbulence. This study demonstrates that the present LES based CFD model can be used as a design tool to remediate problems at existing pump intakes or to test new design ideas and can at least partially replace expensive scaled-model studies.

1.2 Background

Intake structures with one or more pumps are used to withdraw water from a river or a reservoir which is needed for irrigation, domestic and industrial use or in cooling systems of the electric power generation plants. A water intake structure includes an approach channel leading from the water source to a pump column which usually has an entrance structure called the suction or pump bell. The approach channel may also include various flow training devices such as floor and side wall splitters, flow dividers and vanes directing the flow. It is commonly observed that water pumps are subjected to various operational problems such as vibrations and cavitation. All these phenomena result in a significant loss in the efficiency of the pumps. The flow inside a pump intake is generally characterized by the presence of vortices in the vicinity of the pump column. These vortices can be classified as free surface or subsurface vortices according to the locations where they originate, i.e. a vortex is called a free surface vortex if it forms at the free surface of the pump bay. Air entrainment by free surface vortices and ingestion of debris and sediment by floor attached vortices are other examples of problems that adversely affect the pump operation and efficiency.
Two main reasons for the generation of these vortices are poor design of the intake bay (the geometric parameters that define the intake bay are the shape and size of the approach channel and pump column, the clearances between the pump column and the lateral, back and bottom walls of the channel), and the level of non-uniformity of the approach flow. High levels of circulation inside the pump bay usually results in the formation of strong vortices around the pump column. However, some vortices may still be observed in the intake even if the approach flow is uniform. So the goal of the design/redesign process is not so much to eliminate these vortices, but rather to reduce their intensity such that they can not entrain air or debris and the level of swirl inside the pump column is maintained below a threshold value such that the efficiency of the pumps is not significantly affected.

Identification of problems at pump intakes and recommendation of possible solutions to them is usually done by conducting laboratory experiments on scaled models. Such studies helped to understand the physics of the flow at pump bays and to propose solutions for some site specific cases. A disadvantage of experiments conducted on scaled models is that they are expensive and time consuming. Furthermore they do not lead to general criteria for design and cannot account for scaling effects between the model and the prototype. On the other hand, CFD has the potential to predict such flows faster and less costly than the scaled model experiments provided that they can accurately simulate the main physical phenomena present in pump intakes.

Figure 1.1 shows a general view of a simplified (generic) pump bay. The main geometrical parameters of the pump bay are marked in the same figure, where D is the outer diameter of the pipe, L is the distance from the inlet of the approach channel to the back wall of the intake, L1 and L2 are the side wall clearances of the pump column, C is the floor clearance, X is the back wall clearance and S is the submergence depth of the pump column. Satisfactory performance of a pump intake under a certain range of operating conditions requires an appropriate design selection of these geometrical parameters. Additionally, in realistic geometry pump intakes, flow training devices such as pump bells, floor cones, splitter walls, etc. are used. However, the clear effect of these devices on the flow is not well distinguished (e.g. in some cases a splitter wall can
diminish the intensity of the vortical structures in the flow while in others it can amplify it).

1.3 Scope and Objectives of the Present Study
The goal of the present work is to show that advanced CFD models using eddy resolving techniques (LES) and non-dissipative solvers have reached a level of maturity where they can accurately predict the complex flow phenomena present in pump intake structures.

The main objectives of this study are:
1) to validate the proposed LES model by comparing simulation results with recent PIV data collected on a laboratory model of a pressurized pump intake.
2) to compare the performance of the LES model with that of a state of the art commercial RANS code that is used by the industry. To make the comparison as relevant as possible, the equations in the SST RANS model were integrated up to the wall and the mesh was very fine, comparable to that used in LES.
3) to study the flow physics. To achieve several methods are employed to educe and visualize the main vortices present in the pump intake. The LES data sets are used to understand interaction among dominant most energetic coherent structures, to analyze the spectral content of the flow in the vicinity of the pump column and the unsteady swirl distribution inside the pump column.

The long term goal of the present study is to show that LES can be a viable alternative to the experimental and/or RANS based CFD studies that are currently used by the industry to address problems at pump intakes. To achieve this goal the time needed to obtain a solution including statistics using the LES solver should be further decreased.

1.4 Review of Literature

1.4.1 Review of Relevant Experimental Studies
There are numerous experimental studies conducted on scaled models available in the literature because of the need to address problems related to formation of free surface and subsurface vortices and presence of high swirl inside the pump column case by case.
In the following section a summary of some of these studies is given and subsequently, three most recent comprehensive experimental studies are reviewed in detail.

1.4.1.1 Previous Experimental Studies

1.4.1.1.1 Free Surface Vortices

It is commonly observed that free surface vortices are intermittent, meander and under specific conditions can entrain air. In many instances these vortices are the main cause of the problems observed at pump bays. The formation of the free surface vortices depends on the geometrical flow parameters such as the submergence depth of the pump column beneath the free surface and flow conditions (e.g., level of swirl in approach flow, etc.). In this regard one of the parameters that is of great importance and that is usually determined experimentally is the critical submergence defined as the depth at which the free surface vortex begins entraining air bubbles. For instance, Denny (1956) studied the effect of swirl in the incoming flow on the critical submergence. He found that the strength of the vortices diminished with decreasing the back wall or side wall clearances. Similar conclusions were reached by Hecker (1987) and Anwar and Amphlett (1980). Denny suggested that using a pump bell at the entrance of the pump column reduced the separation inside the pump column by making the flow more uniform. However it was concluded that increasing the bell mouth diameter did not affect the critical submergence depth significantly. Experimental work of Bortzsonyi and Kajdi (1975) showed an increase in the strength of the vortex as the back wall clearance was increased. Triplett et al. (1988) confirmed the results of Bortzsonyi and Kajdi (1975). They further found that the floor clearance and bay width did not affect the critical submergence depth significantly.

Quick (1970) based on his observations of the flow at intakes with horizontal pipe intakes suggested boundary shear and vortex stretching as the two main mechanisms responsible for generation of vorticity inside the pump bay. He proposed a criterion for the formation of free surface vortices as a function of the value of the submergence Froude number. Based on his experimental visualizations he concluded that the air entraining vortices were unsteady and intermittent in most of the cases. Chang (1980) conducted a similar study using vertical pipe intakes. He suggested that another
important vorticity source is due to the growth and separation of the boundary layer on the exterior of the suction pipe.

Other experimental studies focused on the correlation of geometrical flow parameters with the critical submergence and the circulation in the approach flow (estimated via some kind of integral scalar quantity). For instance, an empirical relationship between circulation in the approach flow and critical submergence was proposed by Knauss (1987). Jain et al. (1978) found that the critical submergence increases with increasing Reynolds number. They proposed an empirical formulation for the critical submergence as a function of the circulation number, the Froude and Reynolds numbers that defined with the total depth in the channel. The two main factors that determine the critical submergence were suggested as the Froude number and the level of circulation in the approach flow.

The effect of Reynolds number on free surface vortices was investigated, among others, by Anwar et al. (1978). They studied the effect of viscosity on pump intake flows and concluded that it can be neglected for $Re_s > 10,000$-$70,000$ or $Re/Fr_D > 50,000$, where $Re_s$ is the Reynolds number defined with the submergence depth and the critical discharge of the pump intake, and $Fr_D$ is the Froude number defined with the pipe diameter and the mean velocity in the pump column. Zielinski and Villemonte (1968), Daggett and Keulegan (1972), Padmanabhan and Hecker (1984) achieved similar conclusions to those of Anwar et al. (1978). Another study by Anwar (1966) suggested that the free surface tension may be neglected for Weber numbers greater than 120 to 1000 for intakes that have a free surface. Anwar and Amphlett (1980), Padmanabhan and Hecker (1984) and Odgaard (1986) confirmed the observation of Anwar et al. (1978) for flows characterized by high Weber numbers. Padmanabhan and Hecker (1981) investigated the Reynolds number effect on laboratory models. They concluded that testing the models with higher than Froude scaled velocities was not necessary. They further recommended the use of models with geometric scales as large as possible to minimize scaling effects. Recently, Johansson et al. (2005) recommended, based also on the recent revisions of the Hydraulic Institute Standards, the Reynolds (defined with the pump column mean velocity and diameter) and Weber numbers in the model should be above 60,000 and 240, respectively, to avoid significant scale effects. However, in some
cases it was observed that scale effects were present between model and prototype even if the design of the model respected the usual recommendations.

1.4.1.1.2 Subsurface Vortices

Subsurface vortices are the vortices originating from one of the walls of the intake and they are named after their point of origin, e.g. floor attached or side wall attached vortices. Observations of Hecker (1987) suggested that subsurface vortices can be as damaging to pump components as free surface vortices. The subsurface vortices can take the form of weak swirls or they can evolve into very coherent and energetic structures. The common observation is that they form when the circulation in the approach flow is increased over a certain value or the wall clearances are decreased below a certain value.

It is generally accepted that a decrease in the distance between the floor and the pump bell increases the intensity of the floor attached vortices due to higher shear present near the pump bell. Similarly, the reduction of back wall and side wall clearances has generally the effect of strengthening the subsurface vortices originating from these walls.

Quick (1970) concluded that shear in the wall boundary layer and the vortex stretching near the pipe entry region are the main causes of subsurface vortex generation. The factors affecting the formation of the subsurface vortices were also investigated by Tagamori and Ueda (1991). They found that the flow and the vortex patterns near the suction bell depend on the pipe intake clearances from the back wall and side walls.

They observed that the floor clearance significantly affected vortex formation from the surrounding walls. They also concluded that subsurface vortices are generated on the back wall when the back wall clearance is small and on the side walls when the back wall clearance is high.

1.4.1.1.3 Turbulent Swirling Pipe Flow

Free surface and subsurface vortices cause entrance losses, enhanced separation from the pump column, and swirling motions inside the pipe which are, in most cases, the main engineering concern related to pump intake design (Hecker 1987). In pump intake model studies it is common to measure the amount of swirl inside the pump column to determine some measure of the intensity of vortex flow near and inside the pipe. The
decay of the swirl along the pipe was also computed to determine possible effects of swirl on the performance of the pump located downstream in the pump column.

Sayre and Baker (1967) investigated the decay of swirl in smooth pipes with Reynolds numbers between 12,500 and 200,000. Their results showed that the angular momentum flux decreased exponentially along the pipe. This decay rate was found to be a function of Reynolds number and initial conditions. Ito et al. (1979) analyzed the swirl inside a circular pipe. They suggested an empirical equation for the decrease of the circulation inside the pipe.

Padmanabhan and Janek (1980) evaluated the swirl and its effect on the wall pressure for various swirl levels and for several pipe Reynolds numbers ranging from 100,000 to 250,000. They concluded that the swirl decay rate depends on the pipe Reynolds number and it decreases with increasing initial swirl levels. Despite that several studies were conducted on turbulent swirling pipe flows only the study by Padmanabhan and Janek (1980) is directly related to the swirling flow in suction pipes of water intakes. Johansson et al. (2005) found that in some cases intermittent and direction changing swirl may be more harmful to the pump performance than a steady one directional swirl of similar magnitude. One should also mention the extensive review study by Melville et al. (1994) that lists most of the flow related problems at pump intakes along with the improbable cause and design recommendations.

1.4.1.2 The Experimental Study by Ansar (1997)

The study by Ansar (1997) (see also Ansar and Nakato, 1999) aimed to achieve a more basic understanding of pump-approach flow distributions at water intakes. He studied how different geometrical and flow parameters/conditions affect the formation of vortices in the intake flows. He used experimental, numerical and analytical approaches in his study. The experimental study was designed to determine the effect of cross-flow on pump approach flow distributions, and subsequently its effects on the formation of vortices near the vertical pump columns. His experimental model contained a single pump intake with a regular straight approach channel (see Figure 1.2) in which the pump column was placed middle distance between the lateral walls. The overall experimental setup including the channel from which water was withdrawn (perpendicular to the
approach channel) for the case of approach flow with no cross flow is given in Figure 1.3 and the experimental setup for the case of approach flow with cross flow is given in Figure 1.4. The measurements of three velocity components in the approach channel were obtained using Acoustic Doppler Velocimeter (ADV), and axial velocity distributions inside the suction pipe were obtained using Laser Doppler Velocimeter (LDV) techniques.

He concluded that in the case of approach flow without cross flow the mean flow was practically symmetric with respect to the centerline of the intake. Observations showed that high streamwise velocities were present at lower depths along the longitudinal axis of symmetry of the intake (see Figure 1.5 in which a sketch summarizing his measurement results is shown) due to the water being convected into the pump sump. Relatively weak free surface and subsurface (side wall and floor attached) vortices near the suction pipe were observed. In the cross flow case a strong streamwise flow was found to develop at higher depths along one of the side walls and a strong reverse flow was present at lower depths along the right side wall (see Figure 1.6). This causes a large circulation in the pump bay area. He observed two strong vortices near the suction pipe, namely a floor attached vortex beneath the suction bell and a free surface vortex between the suction pipe and the left side wall. Another weak free surface vortex was observed near the right side wall. It was found that the time averaged axial velocity distribution inside the suction pipe was axisymmetric and close to a parabolic shape for the case with no cross flow as shown in Figure 1.7. However, for the cross flow case, the distribution was found to be asymmetric and triangular in shape (see Figure 1.8). Furthermore, in the cross flow case, it was observed that more flow entered the pump column from the back side of the pump bell than from the front.

1.4.1.3 The Experimental Study by Rajendran (1998)

Rajendran (1998) (see also Rajendran and Patel, 2000) conducted an experimental study focusing on the fundamental phenomena responsible for the formation of the main types of vortices in a simplified model of a generic single bay pump intake. The model was operated over a range of Froude numbers obtained by altering the intake velocity. The effect of geometric conditions was tested by varying the clearances between the
intake pipe and the walls and floor of the pump bay. This parametric study investigated the generation of air-entraining free surface vortices and the dependency of critical submergence on the Froude number for different geometric conditions. The results of the study did not fully agree with any set of guidelines typically used in design. The critical submergence values that were found in the study by Rajendran were lower than the suggested values by the guidelines. Free surface vortices were found to form more likely with an increase in the back wall clearance. However, the free surface vortex formation was found to be less affected by the floor clearance than previous studies suggested. Four geometric conditions with different wall clearances were studied. He also conducted experiments with a strongly non-uniform approach flow in order to observe the effect of non-uniformity in the formation of the vortices.

Separate experiments were conducted on a simplified intake (no pump sump) to obtain detailed quantitative validation data for CFD models using PIV techniques. This comprehensive data set was used by Constantinescu (1998) and then by Li et al. (2001) for validation of their RANS models. The experimental setup of the simplified intake is shown in Figure 1.9. It was found that a variety of subsurface vortices occurred simultaneously even at a high submergence depth. A sketch of the main vortices observed in the experiments of Rajendran is shown in Figure 1.10. These vortices were found to be attached to the back wall, side walls and to the floor. Fluctuations in their strength in time was measured (see Figure 1.11) and the vortices were found to meander along the walls they were attached.

At critical submergence, Rajendran observed the presence of two main free surface vortices. They were found to be very unsteady and intermittent. Besides collecting validation data, the study by Rajendran tried to characterize the dynamics of these vortices and the change in their structure in time. To educe these vortices besides PIV, Rajendran (1998) used Laser Induced Fluorescence (LIF) to visualize the eddies present in cross sections of the core of the main vortices and their rotational direction. For instance LIF pictures of the floor attached and the side wall attached vortices observed in the flow are given in Figure 1.12 and Figure 1.13, respectively. PIV was used to obtain quantitative measurements including the number, size, shape, strength and to describe the meandering of the vortices. In the case of small back wall clearance and a
symmetrically positioned pump column, a pair of counter rotating vortices was observed to originate at the back wall. However, in the case of a non-symmetrically placed pump column, only one stronger vortex was observed to form at the back wall. The back wall vortices were found to disappear with increased clearance. Any shift of pump column was found to cause a shift of the vortices on the back wall and floor.

For all four conditions the strongest vortex was found to be the floor attached one. The instantaneous and mean velocity vectors and the out-of-plane vorticity contours in a plane cutting through the floor attached vortex are shown in Figure 1.14 and Figure 1.15, respectively. The meandering of floor attached vortex is described by following the center of the vortex core from the instantaneous PIV frames. The results are shown in Figure 1.16. Rajendran (1998) found that there was no correlation between the meandering velocity and the strength of the vortex. An increase in the floor clearance was found to weaken the floor attached vortex. A similar conclusion was reached for the side wall and back wall vortices. The asymmetric placement of the pump column was found to amplify the value of critical submergence. The sizes of the main free surface vortices were found to be larger than those of the subsurface vortices due to air entrainment. The mean velocity vectors and out-of-plane vorticity contours in a plane near the free surface centered around one of the free surface vortices are shown in Figure 1.17. The free surface vortices were found to meander stronger than the subsurface vortices. A sample result for the previously described free surface vortex is shown in Figure 1.18. The floor and the back wall vortices were found to be dragged to the low velocity side of the flow in the case in which approach flow was non-uniform. The strengths of the back wall and floor attached vortices were found to increase while the side wall attached vortices were found to weaken when the non-uniformity in the approach flow was increased. Rajendran found that the asymmetric placement of the pipe and non-uniform approach flow conditions severely increased the swirl inside the pump column.

1.4.1.4 The Experimental Study by Yulin et al. (2000)

Yulin et al. (2000) studied experimentally using PIV techniques the flow in a pressurized pump intake (no free surface) containing several flow training devices. The
study aimed to investigate the flow features in the vicinity of the pump bell and to provide a compressive data set for validation of numerical models.

The laboratory model included an irregular rectangular water pump sump, 2500mm in length, 400mm wide, and 248mm deep with a single vertical intake pipe (see Figure 1.19). The intake had two inlet channels (inlet 1 and inlet 2) with identical cross sections divided by a splitter wall. The leading side of the splitter wall was semicircular and its section varied in the streamwise direction. The purpose of using two inlets was stated as to create the asymmetric flows around the pump bell which made the experiments more realistic. Trash racks were placed in front of each inlet to stabilize the flow. The shape of the pump column near the pump sump was designed to reduce separation and minimize swirl inside the pump column. Figure 1.20 shows the structure of the pump bell. All the main dimensions (in millimeters) of the bell and intake are given in Figure 1.20 and Figure 1.21.

In the experiments, five different operation conditions were considered based on the mean velocity at the two inlets. The total pump outflow rate varied between 1.60 m³/min and 2.40 m³/min and the ratio between inlet 1 and inlet 2 were respectively 1.22, 1.5, 2.35, 4.0 and 1.0. Out of these operation conditions, case 3 was selected for the validation test case in the present numerical study. The data obtained from the experiments included the flow rate in both of the inlet channels, the water temperature, the static pressure at certain representative points upstream of the sump and the air content ratio of the water. It also included the data obtained by PIV techniques namely, the time averaged in plane velocity components (unfortunately, the third component was not available), the root mean square of the velocities and the turbulent kinetic energy (two velocity components only) at the measured cross sections. The experiments also showed that by using the very particular pump sump shown in Figure 1.20, the cavitation was reduced and the efficiency of the pump station was enhanced. No attempt was done to study or characterize the changes of the structure and intensity of the main vortices in time or to describe the global vortical structure of the flow.
1.4.2 Review of the Numerical Studies

In contrast to the large number of experimental studies available on pump intake flow and design, numerical investigations of such flows are scarce. In the following section a summary of some of the most relevant numerical studies is given.

1.4.2.1 Previous Numerical Studies

Tagomori and Gotoh (1989) employed a finite volume method to solve the RANS equations in conjunction with the standard k-ε turbulence model. They studied the effect of non-uniformity in the approach flow on the generation of various vortices. They also investigated the effectiveness of various flow training devices. To suppress the vortices forming near the side walls, they added a step to the back wall and a curtain wall upstream of the pump bell. The effects of a flow splitter attached to the back wall and changing levels of free surface elevation on the side wall attached vortex was investigated. The experimental flow visualizations accompanying their numerical results suggested that their simulations were qualitatively successful at least in some respects. Unfortunately they did not provide any information on their methodology, grid generation, grid sizes, etc.

Takata et al. (1992) was the first to use LES to simulate flow in a pump intake at Reynolds numbers between $1.5 \times 10^4$ and $5.5 \times 10^4$. They employed the standard Smagorinsky model with a constant coefficient to relate the sub-grid stresses to the rate of strain of the resolvable scales and wall functions. The governing equations were discretized in space using second order accurate finite difference schemes. Integration in time was achieved using an explicit time marching scheme. The calculations were carried out in a model intake with a vertical splitter wall in the plane of symmetry. The effect of the swirl due to unequal velocities in the two sections of the approach flow was investigated. The grid used in their study had around 252,000 cells, which was too coarse to obtain accurate results in LES. Therefore their results were only providing a qualitative description of the flow.

Lu et al. (1997) solved the RANS equations in conjunction with the standard k-ε model and wall functions. They investigated the effect of wall clearances and pump bell shape on the level of velocity distribution uniformity inside the pump column and bay.
Based on their simulations they recommended an “optimum hydraulic criterion” for intakes. They suggested optimum values for the floor clearance, intake bay width, back wall clearance, bay length and bell mouth diameter. Their recommendations were close to the findings of previous experimental studies. Unfortunately, they did not provide specific information on the Reynolds numbers in the computations, their numerical method, etc.

1.4.2.2 The Numerical Study by Ansar (1997)

Ansar (1997) (see also Ansar et al., 2002) conducted a numerical study of two pump intake geometries, one containing a single pump column (see Figure 1.21) and the other containing two pump columns (see Figure 1.22). He used an inviscid model (the solid surfaces were treated as slip walls) implemented into the same solver used by Constantinescu (1998) under the assumption that the main cause of vortex formation is due to inviscid rotational effects in the intake. His work aimed to determine the reliability of a simple inviscid model (no turbulence model) in predicting the flow structures in the approach flow channel of water intakes and the formation of vortices near the suction pipe by comparing the numerical results with the experimental data collected on a model pump intake part of the same study.

He found that the inviscid model was able to predict reasonably well the velocity profiles in the approach channel away from the solid boundaries for both the cross flow and the no cross flow cases. Also the main subsurface vortices observed in the experiments were captured by the numerical simulations. The direction of the rotation of the subsurface vortices was also correctly predicted. Though the free surface flow pattern was reasonably well predicted, some disagreement existed in the prediction of the location of some of the free surface vortices. In the simulation containing two pump columns, the dividing streamline was found to be along the symmetry axis in the case of equal discharges in the two pump columns as shown in Figure 1.23a and Figure 1.23b. When there was an uneven split of discharges between the pump columns (7-to-3 ratio), the dividing streamlines were found to shift toward the side wall adjacent to the pipe carrying the smaller discharge, as one approaches the sump floor (see Figure 1.23c and Figure 1.23d). This flow imbalance also resulted in the strengthening of the intensity of
the vorticity in the pump columns region. Froude number and submergence depth was found to significantly affect the free surface vortices formed near the suction pipe.

1.4.2.3 The Numerical Study by Constantinescu (1998)

The aim of the study by Constantinescu (1988) (see also Constantinescu and Patel, 1998 and 2000, Rajendran et al., 1999) was to develop a RANS based model with near-wall modeling capabilities that can be used to improve design methods to avoid the formation of strong free surface and subsurface vortices in pump intakes. Their study concentrated on a simplified pump intake geometry (see Figure 1.24) which did not contain the usual vortex suppressing devices found in realistic intakes but in which the flow retained the main complexities encountered in more complex geometries. The numerical method was shown to be robust and efficient on highly stretched, large aspect ratio grids for simulations at high Reynolds numbers. The viscous solver used a fully implicit, fractional step algorithm in conjunction with ADI and an approximate factorization scheme to solve the momentum, pressure and turbulence model equations in generalized curvilinear coordinates. Part of the study (Constantinescu and Patel, 2000) investigated the effect of the turbulence model and the effect of wall roughness (introduced in the model via the boundary condition for $\omega$ in the k-\omega model) on the flow, e.g. the differences in the prediction of critical flow features, such as subsurface and free surface vortices and the level of swirl inside the pump column were quantified. A main conclusion of the study was that a near wall turbulence model is essential to predict the important flow features in pump intakes. This is because large adverse pressure gradients and massive separation are present in the pump column region. He concluded that the two layer k-\epsilon model and the low Reynolds number k-\omega model predict similar results on smooth surfaces (see Figure 1.25 and Figure 1.26). It was found that the high Reynolds number k-\omega model, which takes roughness into account, predicted weaker vortices. This result showed that the effect of roughness was simulated correctly, at least qualitatively. Based on this observation, it was suggested that the wall roughness can be used as a vortex suppressing device. The model was validated using the experimental data (which included mean velocity and vorticity distributions in relevant sections measured using PIV techniques) collected by Rajendran (1998) in an intake of identical geometry (see
Figure 1.24) and for same flow conditions. The grid used in the validation contained around 550,000 cells. The Reynolds number of the flow inside the pump column was 45,000. It was found that the number and the type of the vortices observed in the experiments were predicted correctly by the model. For instance, Figure 1.27 and Figure 1.28 show both qualitative and quantitative comparison between simulation and experiment for the flow in planes cutting through the cores of the floor attached vortex and side wall attached vortex, respectively. The distribution of the vorticity inside the cores of these vortices was also predicted reasonably well by the model, especially for the vortices that were found to be relatively stable in the experiment. The main vortices were visualized with different methods including three dimensional streamlines, an example showing one of the main free surface vortices and a corner vortex is shown in Figure 1.29. Simulations showed that for Re>50,000 the overall vortical structure of the flow does not change significantly. The effect of incoming circulation in the approach flow was also investigated. It was found that the size, number, intensity of the vortices and swirl inside the pump column was significantly affected by the level of non-uniformity in the approach flow. It was found that the asymmetric location of the pump column and the presence of shear in the upstream part of the approach channel tend to amplify the vortices and change their structure. In agreement with previous experimental studies, the simulations showed that the intensity of the back wall and the floor attached vortices strengthen when the back wall and floor clearances were decreased. The decrease of the submergence depth was found to intensify the free surface and subsurface vortices. The main deficiency of the model was that it was not easily adaptable to cases in which the pump intake geometry would contain vortex suppressing devices of complex geometry.

1.4.2.4 The Numerical Study by Li et al. (2001)

In the study by Li et al. (2001) (see also Li et al., 2004) a general CFD RANS code that can use structured or unstructured multi-block meshes, U²RANS, was used to simulate three test cases of different complexity including the flow in a pump intake of very complex geometry.

The first validation study was done for the same simplified intake model geometry used in the validation study by Constantinescu (1998) (see Figure 1.24).
Simulation was conducted on a grid with 330,000 cells. The standard k-ε turbulence model with wall functions was used. For the simple intake case, all the mean vortices observed in the experiments were accurately simulated, in the sense that the shape, location and direction of rotation of the main eddies were predicted correctly. Overall the results obtained with U²RANS showed the same level of agreement with the experimental data as the previous simulation of Constantinescu (1998). A sample of the validation results in a plane cutting through the floor attached vortex is shown in Figure 1.30. Two U²RANS solutions were calculated. Figure 1.30c shows the case for which the wall thickness of the pipe was considered and Figure 1.30d shows the case for which the wall thickness of the pipe was not considered (NBNT stands for “no bell and no wall thickness”). With the exception of the free surface vortices, the pipe wall thickness was found to have insignificant effect on the vortex patterns inside the pump bay.

The second validation case was for a slightly more complex intake geometry in which the pump column started with a pump bell (see Figure 1.2 and Figure 1.31 where the position of the planes in which velocity profiles were measured in the experiment is shown). Two approach flow conditions were simulated, one with cross flow and the other without cross flow similar to the cases studied experimentally in the study by Ansar (1997). The grids used in the simulations contained around 480,000 cells for both cross flow and no cross flow cases. The physical Reynolds number of the flow in the pump column was approximately 120,000. The mean flow features in the approach channel and near the pump column including the streamwise velocity distribution in the approach channel and decay of the swirl inside the pump column were compared with the results of Ansar (1997). The comparison of numerical and experimental results for the streamwise velocities in the no cross flow and cross flow cases are shown in Figure 1.32 and Figure 1.33, respectively. The axial velocity near the pump throat was found to be in good agreement with experimental results for the no cross flow case. However, in some regions, the model underestimated the velocity magnitudes. In the case with cross flow, numerical results suggested a more uniform distribution than the experimentally measured V shaped distribution for the axial velocity at the pump throat. Numerical results confirmed that the swirl inside the pump column in the case with no cross flow
was very weak. As expected, the approach flow in the case with cross flow caused a strong swirl inside the pipe intake consistent with experimental results.

Finally, U²RANS was applied to the Red Hills Generating Facility Circulating Water Pumps in Mississippi project (Figure 1.34) for which the experimental model study was conducted by Nakato et al. (1999). The grid employed in this simulation contained around 1,000,000 cells and the physical Reynolds number of the flow was approximately 370,000. The effect of inlet flow and the bell shape on the swirl inside the pump column was investigated and the velocity distribution at the pump throat was evaluated. The simulations of the practical pump intake, one including the far field (Figure 1.34) and one concentrating on the flow details in the pump bay (Figure 1.35), showed that the flow distribution at the bay entry was very non-uniform due to the flow structures present upstream of the bay. These findings were also supported by the experimental results. As an example of the results obtained from these simulations, the approach flow observed in the experiment is shown together with two dimensional streamlines from the numerical results in a longitudinal section cutting through the symmetry axis of the pump column in Figure 1.36. This figure suggests that the flow patterns were predicted correctly by the U²RANS simulation, at least qualitatively. Both experimental and numerical results predicted a very weak swirl inside the pump column under symmetric and asymmetric operating conditions. The U shaped velocity distribution at the pump throat observed in the experiments was also predicted numerically.

The general conclusion was that the study by Li et al. (2001) illustrated the robustness of U²RANS as a cost effective tool for pump intake design. However, the steady RANS model was not able to capture any of the unsteady phenomena that characterize the flow in pump intake geometries and for the intakes of complex geometry the amount of experimental data was insufficient to conduct a detailed quantitative validation study.
Figure 1.1 General view of a simplified pump intake configuration used in the study by Constantinescu and Patel (1998)
Figure 1.2 General view of the pump intake geometry described in the study by Ansar (1997)
Figure 1.3 Experimental setup of Ansar et al. (1997) for approach flow with no-cross flow case
Figure 1.4 Experimental setup of Ansar et al. (1997) for approach flow with cross flow case
Figure 1.5 Sketch of the observed approach flow distribution for the no-cross flow from the work of Ansar et al. (1997)

Figure 1.6 Sketch of the observed approach flow distributions for the cross flow from the work of Ansar (1997)
Figure 1.7 Axial velocity distribution at the pump throat for the no cross flow case in the study by Ansar (1997)

Figure 1.8 Axial velocity distribution at the pump throat for the cross flow case in the study by Ansar (1997)
Figure 1.9 Experimental setup of the pump bay used in the study by Rajendran (1998)

Figure 1.10 Sketch of vortices observed in the model pump intake in study by Rajendran (1998)
Figure 1.11 Variation of instantaneous strength of each vortex as a function of time measured by Rajendran (1998)

Figure 1.12 Visualization of the floor attached vortex using LIF in the experiments of Rajendran (1998). Picture is taken from the channel bottom
Figure 1.13 Visualization of side wall 1 vortex by LIF in the experiments of Rajendran (1998)

Figure 1.14 Instantaneous velocity vectors and out-of-plane vorticity contours in a plane parallel to the bottom cutting through the floor attached vortex from the experimental study by Rajendran (1998)
Figure 1.15 Average velocity vectors and out-of-plane vorticity contours in a plane parallel to the channel bottom cutting through the floor attached vortex from the experimental study by Rajendran (1998)

Figure 1.16 Meandering of floor attached vortex measured by Rajendran (1998) [Pipe column projection on the floor is shown with dashed line]
Figure 1.17 Average velocity vectors and out-of-plane mean vorticity contours in a plane situated near the free surface showing the free surface vortices from the experimental study by Rajendran (1998)

Figure 1.18 Meandering of the free surface vortex measured by Rajendran (1998)
Figure 1.19 General view of the pump intake geometry considered in the experiments of Yulin et al. (2000)
Figure 1.20 Detailed schematic of the pump bell of the pump intake geometry considered in the experiments of Yulin et al. (2000)
Figure 1.21 Computational grid for the single pump intake simulated in the numerical study by Ansar (1997)
Figure 1.22 Computational grid for the pump intake geometry containing two pump columns simulated by Ansar (1997)
Figure 1.23 Streamlines patterns calculated by Ansar (1997) in the pump intake containing two pump columns; a) Streamlines at the free surface in the case of equal discharges in the pumps (Q1 = Q2); b) Streamlines in plane situated between floor and pump bell level showing the floor attached vortices in the case of equal discharges in the pumps (Q1 = Q2); c) Streamlines at the free surface in the case of unevenly split discharges in the pumps (Q1 > Q2); d) Streamlines in plane situated between floor and pump bell level showing the floor attached vortices in the case of unevenly split discharges in the pumps (Q1 > Q2)
Figure 1.24 Intake configuration used in the validation study by Constantinescu (1998)
Figure 1.25 Streamlines at the free surface from the study by Constantinescu (1998)

Figure 1.26 Visualization of free surface vortices using vorticity contours from the study by Constantinescu (1998)
Figure 1.27 Comparison of the floor attached vortex structure (streamlines and out of plane vorticity from experiments and simulations; a) general flow pattern at a plane situated between the floor and the pump column entry level; b) numerical simulations of Constantinescu (1998); c) experimental results of Rajendran (1998)
Figure 1.28 Comparison of the side wall attached vortex structure (streamlines and out of plane vorticity from experiments and simulations; a) general flow pattern at a plane situated between side wall 2 and the pump column; b) numerical simulations of Constantinescu (1998); c) experimental results of Rajendran (1998)
Figure 1.29 Visualization of; a) main free surface vortex; b) corner vortex using particle path lines from the simulations of Constantinescu (1998)
Figure 1.30 Comparison between the streamlines of the floor attached vortex obtained from the numerical simulations of Li et al. (2001) and Constantinescu (1998) and the experimental results of Rajendran (1998)
Figure 1.31 Location of the velocity measurement stations in the approach channel in the study by Li et al. (2001). The pump column contains a pump bell similar to the experiment of Ansar (1997).
Figure 1.32 Comparison of the streamwise velocity profiles obtained from the simulations of Li et al. (2001) and the experiments of Ansar (1997) for the case of no cross flow at sections B, C and D (see Figure 1.31)
Figure 1.33 Comparison of the streamwise velocity profiles obtained from the simulations of Li et al. (2001) and the experiments of Ansar (1997) for the case with cross flow at sections B, C and D (see Figure 1.31)
Figure 1.34 Configuration of the pump intake geometry, including the far field, simulated in the study by Li et al. (2001)

Figure 1.35 Configuration of the pump intake geometry (near field) simulated in the study by Li et al. (2001)
Figure 1.36 Qualitative comparison between the numerical simulations and the experiments from the study by Li et al. (2001) in a longitudinal plane cutting through the symmetry axis of the pump column.
CHAPTER 2. NUMERICAL METHODS

2.1 Introduction
Two different numerical methods were employed to simulate the flow in a pump intake of realistic geometry. First, the mean flow was simulated using a steady state Reynolds Average Navier-Stokes (RANS) model implemented into a commercial code, Fluent. Second, a time accurate LES model in which the model coefficient is estimated dynamically was used to predict the flow and dynamics of vortices inside the same intake configuration. The details of these models are given in the next sections.

2.2 Reynolds Averaged Navier-Stokes Model
RANS based numerical simulations were performed using Fluent, a general CFD solver for modeling fluid flow and heat transfer in complex geometries. Fluent is a state-of-the-art widely used commercial package for applications in many areas of engineering. The numerical method in Fluent is based on a cell-centered finite volume method in conjunction with a linear reconstruction scheme capable of using both structured and unstructured meshes which provides complete mesh flexibility needed in complex topologies. As a preprocessor for geometry modeling and mesh generation, Gambit® was used. Gambit allows the user to generate high quality meshes using unstructured multi-block grids. Fluent can use a wide range of turbulence models with both wall functions and near-wall treatment to simulate turbulent flows.

For the present study the segregated solver was used. The flow was assumed incompressible. Gradients of the solution variables are computed using Green-Gauss’ theorem. Diffusion terms are discretized using second order central scheme. For the convective terms, there are several choices offered in the code including first order upwind, second order upwind, QUICK and third order MUSCL schemes. For the convective terms the second order upwind scheme was chosen as the discretization scheme in the present simulations. The discretized equations are solved using point wise Gauss-Seidel iteration in conjunction with an algebraic multi-grid method to accelerate the solution convergence.
2.2.1 General Description of SST Model

The shear stress transport (SST) model which can be integrated up to the wall was used as the turbulence model. The SST model uses a k-ω formulation, with the original Wilcox model activated near the walls and the standard k-ε model activated in the outer wall region. Unlike other two-equation RANS models, the k-ω/SST model allows simple Dirichlet boundary conditions to be specified at the wall for the transported turbulence quantities. Cross diffusion terms and blending functions were added to the ω equation typically used in the standard k−ω model. The SST model has a performance very similar to that of the original k-ω model, but without the undesirable free stream dependency. The definition of the eddy viscosity in the SST model is modified to account for the definition of the principal turbulent shear stress. The new model was found to lead to improvements in the prediction of adverse pressure gradient flows. Several studies have shown that SST is one of the most accurate RANS models in predicting complex vortical flows. One should also point out that modifications that allow the treatment of rough walls and surface mass injection are available.

2.2.2 Transport Equations and Parameters for the SST Model

Implemented in Fluent

The SST model is similar to the standard k-ω model for which the equations are given as:

\[ \frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho u_i u_i) = \frac{\partial}{\partial x_j}(\Gamma_k \frac{\partial k}{\partial x_j}) + G_k - Y_k \tag{2.1} \]

\[ \frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho \omega u_i) = \frac{\partial}{\partial x_j}(\Gamma_\omega \frac{\partial \omega}{\partial x_j}) + G_\omega - Y_\omega + D_\omega \tag{2.2} \]

In these equations, \( \rho \) is the density, \( k \) is the turbulent kinetic energy, \( u_i \) is the velocity component in the \( i \) direction, \( t \) is the time and \( \omega \) is the specific dissipation rate. \( G_k \) represents the generation of turbulent kinetic energy due to mean velocity gradients. \( G_\omega \) represents the generation of \( \omega \). \( \Gamma_k \) and \( \Gamma_\omega \) represent the effective diffusivity of \( k \) and \( \omega \), respectively. \( Y_k \) and \( Y_\omega \) are the dissipation of \( k \) and \( \omega \) due to the turbulence. \( D_\omega \) represents the cross diffusion term.
The effective diffusivity for the SST model is given by:

\[ \Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \] (2.3)

\[ \Gamma_w = \mu + \frac{\mu_t}{\sigma_\omega} \] (2.4)

where \( \mu \) is the molecular viscosity of the flow and \( \mu_t \) is the turbulent viscosity. \( \sigma_k \) and \( \sigma_w \) are the turbulent Prandtl numbers for \( k \) and \( \omega \), respectively. The turbulent viscosity \( \mu_t \) is computed as follows:

\[ \mu_t = \frac{\rho k}{\omega} \left\{ \max \left[ \frac{1}{\alpha^*, \frac{\Omega F^*}{\sigma_\omega}} \right] \right\} \] (2.5)

where \( \Omega \) may be computed as

\[ \Omega \equiv \sqrt{2\Omega_{ij} \Omega_{ij}} \] (2.6)

\[ \sigma_w = \frac{1}{F_1 / \sigma_{w1} + (1 - F_1) / \sigma_{w2}} \] (2.7)

\[ \sigma_k = \frac{1}{F_1 / \sigma_{k1} + (1 - F_1) / \sigma_{k2}} \] (2.8)

where \( \Omega_{ij} \) is the mean rate of rotation tensor and the coefficient \( \alpha^* \) is introduced to damp the turbulent viscosity and it acts as a low-Reynolds number correction. It is given by:

\[ \alpha^* = \alpha^*_x \left( \frac{\alpha^*_0 + \text{Re}_i / R_k}{1 + \text{Re}_i / R_k} \right) \] (2.9)

where,

\[ \text{Re}_i = \frac{\rho k}{\mu \omega} \] (2.10)

\[ R_k = 6 \]

\[ \alpha^*_0 = \frac{\beta_i}{3} \]
$\beta_i = 0.072$

For the high Reynolds-number version of the SST model, $\alpha^*$ and $\alpha_\infty^*$ are taken as

1. The blending functions $F_1$ and $F_2$ are given by

$$ F_1 = \tanh(\phi_1^4) $$

$$ \phi_1^4 = \min \left[ \max \left( \sqrt{\frac{k}{0.09 \omega y}}, \frac{500 \mu}{\rho y^2 \omega} \right), \frac{4 \rho k}{\sigma_{\omega,2} D_{\omega}^+ y^2} \right] $$

$$ D_{\omega}^+ = \max \left[ 2 \rho \frac{1}{\sigma_{\omega,2}} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-20} \right] $$

$$ F_2 = \tanh(\phi_2^2) $$

$$ \phi_2^2 = \max \left( 2 \frac{\sqrt{k}}{0.09 \omega y}, \frac{500 \mu}{\rho y^2 \omega} \right) $$

The term $G_k$ represents the production of turbulent kinetic energy, and is defined similarly as in the standard k-\omega model.

$$ G_k = \mu_i S^2 $$

where $S$ is the magnitude of the rate of strain tensor.

The term $G_\omega$ represents the production of $\omega$ and is given by

$$ G_\omega = \frac{\alpha}{\nu_i} G_k $$

Note that this formulation differs from the standard k-\omega model. Another difference between the k-\omega and SST models is in the way the term $\alpha_\infty^*$ is evaluated. In the standard k-\omega model $\alpha_\infty^*$ is defined as a constant, $\alpha_\infty^* = 0.52$. In the SST model, $\alpha_\infty^*$ is given by

$$ \alpha_\infty^* = F_1 \alpha_{\infty,1} + (1 - F_1) \alpha_{\infty,2} $$

$$ \alpha_{\infty,1} = \frac{\beta_{i,1}}{\beta_i^*} - \frac{\kappa^2}{\sigma_{\omega,1} \sqrt{\beta_i^*}} $$

$$ \alpha_{\infty,2} = \frac{\beta_{i,2}}{\beta_i^*} - \frac{\kappa^2}{\sigma_{\omega,2} \sqrt{\beta_i^*}} $$
where $\kappa$ is 0.41 and $\beta_{i,1}$ and $\beta_{i,2}$ are 0.075 and 0.0828 respectively.

The term $Y_k$ represents the dissipation of turbulence kinetic energy.

$$Y_k = \rho \beta^* k \omega$$

(2.21)

The term $Y_\omega$ represents the dissipation of $\omega$.

$$Y_\omega = \rho \beta_4 \omega^2$$

(2.22)

Instead of having a constant value, $\beta_i$ is given by

$$\beta_i = F_i \beta_{i,1} + (1 - F_i) \beta_{i,2}$$

(2.23)

where $\beta_{i,1}$ and $\beta_{i,2}$ are 0.075 and 0.0828 respectively.

To blend the $k$-$\varepsilon$ and $k$-$\omega$ models together, the standard $k$-$\varepsilon$ model equations have been transformed into equations based on $k$ and $\omega$, which leads to the introduction of a cross-diffusion term ($D_\omega$). In the SST formulation $D_\omega$ is defined as

$$D_\omega = 2(1 - F_i) \rho \sigma_{\omega,2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

(2.24)

The model constants are

$$\sigma_{k,1} = 1.176 \quad \sigma_{\omega,1} = 2.0$$

$$\sigma_{k,2} = 1.0 \quad \sigma_{\omega,2} = 1.168$$

$$a_i = 0.31 \quad \beta_{i,1} = 0.075 \quad \beta_{i,2} = 0.0828$$

(2.25)

2.3 Large Eddy Simulation Model

LES has been largely used until recent years to study turbulent flows in simple configurations at relatively low Reynolds numbers. Most predictions of engineering flows are obtained using the RANS approach in which the effect of most of the scales on the mean flow is accounted via a RANS turbulence model. In contrast to that, in LES the dynamically important scales in the flow are directly computed, and only the effect of the filtered scales on the resolved scales is modeled. The main motivation of the use of LES is mainly its greater accuracy compared to RANS, especially its ability to better predict turbulent mixing phenomena which are present in so many applications that are relevant to hydraulics and environmental fluid mechanics, as well as massively separated flows which are dominated by coherent structures (e.g., flow in pump intakes). Unfortunately
the numerical methods used in RANS or even Unsteady RANS codes are not directly applicable to LES. RANS typically uses upwinding schemes to discretize the nonlinear terms or other dissipative formulations, such as adding explicitly numerical dissipation terms in the momentum equations, which provide the numerical dissipation needed to make the numerical algorithm robust. In fact, most of today’s codes used to perform LES for complex engineering codes are just unsteady RANS codes in which a sub-grid stress (SGS) model has replaced the RANS model. This approach is however not suitable for obtaining accurate LES solutions (especially when the flow is dominated by coherent structures that contain a relative high amount of energy over a large spectrum of frequencies), since the numerical dissipation present in these codes competes and in some cases is larger than the dissipation introduced by the SGS model. As opposed to Direct Numerical Simulation (DNS), in LES the dissipative scales of motion are not resolved by the grid, thus straight-forward use of non-dissipative (central-differences) discretizations leads to numerical instability. The solution to this fundamental problem related to robust LES methods for complex geometries with non-dissipative schemes is to develop non-dissipative schemes that discretely conserve kinetic energy (Arakawa, 1966, Perot, 2002, Mahesh, Constantinescu and Moin, 2004).

A massively parallel LES flow solver (Mahesh, Constantinescu and Moin, 2004 and Mahesh et al., 2003) is used in the present work. The code was run on several computing platforms and has shown close to linear scalability (time needed to obtain a solution is inversely proportional to the number of processors) in the range of 4-100 processors. The algorithm, which was developed for unstructured grids, is non-dissipative (central-differences schemes are used to discretize all operators in the governing equations, including the convective terms) yet robust at high Reynolds numbers on highly skewed meshes. No filtering is applied on the resolved velocity field. Discrete energy conservation was found essential to ensure this behavior.

The dynamic Smagorinsky sub-grid scale model (Lilly, 1992) is used to account for turbulence effects. The use of this model as opposed to the classical constant coefficient Smagorinsky model has several important advantages such as eliminating the need to specify the value of the model constant which becomes a function of space and time and is calculated from the resolved variables without any empirical input,
eliminating the need for near-wall corrections (Van Driest damping functions) for the viscosity in the near-wall region, predicting zero SGS viscosity in the laminar regions of the flow (e.g., outside the turbulent boundary layers), eliminating the need to use empirical modifications to account for effect of ‘extra strains’ due to stratification and rotation effects, if present.

A collocated, finite volume scheme on unstructured grids with arbitrary elements is used to solve the filtered Navier-Stokes equations:

\[
\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial \tilde{u}_i \tilde{u}_j}{\partial x_j} = -\frac{\partial \tilde{p}}{\partial x_i} + \frac{1}{\text{Re}} \left( \nu_t + \nu_r \right) \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right)
\]

(2.26)

\[\nu_t = (C\Delta^2) \left| \frac{\partial S}{\partial t} \right| \]

(2.27)

\[\left| \frac{\partial S}{\partial t} \right| = 2 \sqrt{\bar{S}_{ij} \bar{S}_{ij}}
\]

(2.28)

\[\bar{S}_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right)
\]

(2.29)

\[C_d \Delta^2 = \frac{1}{2} < \frac{L_{ik} M_{ik}}{M_{ik}^2} >
\]

(2.30)

\[L_{ik} = -\bar{u}_i \bar{u}_k + \bar{u}_i \bar{u}_k
\]

(2.31)

\[M_{ik} = \left( \frac{\Delta}{\Delta} \right) \left| \bar{S} | \bar{S}_{ik} - | \bar{S} \right| \bar{S}_{ik} \]

(2.32)

where \(\Delta\) is a measure of the local grid spacing and the ratio of the test (\(\sim\)) to grid (\(-\)) filter width was taken equal to 2. The \(<<>\) operator in (2.30) refers to local filtering that replaces the usual averaging done in the homogeneous directions in simple flows. As implicit filtering is used with the dynamic Smagorinsky model, \(\bar{u}_i\) refers to the grid filtered (resolved) velocity component in the ‘i’ direction. The test filter corresponds to an averaging over the neighboring cells. As seen from the model description, there are no adjustable constants in the SGS model.
The code was parallelized using MPI and has showed good scalability on different platforms (e.g., large PC clusters, IBM SP III, Origin 3000) up to couple of hundred of processors allowing simulations with grid sizes in the range of ten million computational cells.

The predictive abilities of the solver were tested for a wide range of flows including isotropic turbulence, turbulent channel flows, turbulent flow over cylinders and spheres (Mahesh, Constantinescu and Moin, 2004), particle laden flow in a coaxial geometry (Apte et al., 2003), spatially evolving round jets (Babu and Mahesh, 2004), jets in crossflow (Muppidi and Mahesh, 2004) and flow in realistic jet aircraft combustor geometries (Mahesh et al., 2005).
CHAPTER 3. DESCRIPTION OF COMPUTATIONAL DOMAIN, MESH, FLOW AND BOUNDARY CONDITIONS

3.1 Introduction
In the present study, a pump intake (Figure 1.1) of identical geometry to the one in the experimental model of Yulin et al. (2000) was used in both RANS and LES simulations. This chapter presents a synopsis of the physical description of the domain, flow and boundary conditions and the grid generation.

3.2 Physical Description of the Domain and Test Conditions
The model employed in this study is a pressurized pump intake, i.e. it does not have a free surface (Figure 3.1). The pump bay consists of one pump column with two approaching channels separated by a divider wall and contains several vortex suppressing devices near the junction of the intake walls and a pump bell of complex geometry which is exactly reproduced in the model. The interior diameter of the downstream part of the intake pipe was taken as the length scale (D=130mm). The width of the channel is b=1.49D, and the height, H, of the inlets is 1.9D. The height and width, c, of the corner fillets are 0.25D. The length, L, of the channel is 7.7D. The back wall of the intake is situated 0.6D from the center of the pump column in the downstream direction and the inlets are situated 7.1D in the upstream direction. The pump column is placed symmetrically between the two side walls. The distance from pipe center to the lateral walls, W₁ and W₂, are equal to 1.54D. The submergence of the pump column, S, is 1.3D. The length of the flow divider, Lₙ, is 5.1D. It has a circular head, starting 0.8D upstream of the pipe center and ending at the inlet section. Its thickness changes from 0.3D to 0.09D over its length. The maximum diameter of the bell mouth is Dₘₙ=1.23D (see Figure 1.20 and Figure 1.21).

Test case 3 from Yulin et al. (2000) which has a large ratio between the discharges of inlet 1 and inlet 2 channels was selected. The discharges through the two inlet sections were 0.905 and 0.385 m³/min, respectively. This imbalance had the effect of formation of a very intense floor attached vortex. The velocity length scale, U, was
taken as the mean velocity inside the pump column \((U \approx 1.6 \text{m}^3/\text{s})\) at a section where the diameter was equal to \(D\). With this non-dimensionalization, the pipe intake Reynolds number is 210,600 while the non-dimensional mean velocities in the two inlet channels are 0.188\(U\) and 0.08\(U\), respectively. The characteristic Reynolds number outside the pump column is only one fifth of the one that characterizes the flow inside the pump column. This flow feature is used when determining the cell sizes in the different regions of the flow domain for LES.

### 3.3 Boundary Conditions

#### 3.3.1 Inlet Sections

The model employed in the present study has two inlet sections. A fully developed turbulent velocity profile was specified at the inlet 1 and inlet 2 sections of the pump intake. These profiles were obtained from preliminary simulations of the flow in straight channels with identical sections to inlet 1 and inlet 2 at the corresponding Reynolds numbers. In these channels, the flow in the streamwise direction was assumed to be periodic.

In the RANS case the calculated profiles included the mean velocity components along with \(k\) and \(\omega\) (same SST model was used). The mesh contained around 200,000 cells.

A similar procedure was applied for LES, however the calculated profiles contained only the three instantaneous velocity components (realistic turbulence). This way we avoided any artificial way of trying to model the turbulence fluctuations (this approach is called synthetic turbulence) and reduced the uncertainty to a minimum in specifying inflow boundary conditions in LES, which is critical for the overall success of the validation. The LES channel simulation for the larger channel discharge necessitated a grid with about 3 million control volumes (first grid point off the wall situated at about \(y^+=0.3\)) and turbulent velocity fields were collected at an arbitrary section over a non-dimensional time of 60\(D/U\) in both simulations. The instantaneous velocity fields were then fed in a time accurate way through the two inlet sections during the simulation of the flow in the pump intake geometry.
3.3.2 Outlet

The specification of the flow variables in the outlet section was obtained using linear extrapolation from the interior of the domain in both LES and RANS. Specifically, a mass outflow boundary was used in the RANS simulations. Additionally, the streamwise velocity component at the outflow was corrected to insure global conservation of mass at each time step (LES) or iteration (RANS).

3.3.3 Walls

No slip velocity conditions were used at the walls, as the mesh was fine enough to resolve the near wall region in both LES and RANS. This avoids the use of wall functions and the approximations associated with this approach (e.g., it is well known that the validity of the law of the wall breaks near solid boundaries in separated flow regions). The walls were assumed to be smooth, thus boundary roughness was not taken into consideration.

3.4 Grid Generation

The quality of the computational mesh has an important role in achieving the desired accuracy of the simulations especially if the computational domain is very complex and eddy resolving techniques coupled with non-dissipative flow solvers are employed. The grid generation was done such that a proper resolution of the wall boundary layers and of the regions around the pump column, where wall attached vortices are present, was achieved.

Grid generation was done using Gambit, which is the grid generator part of the commercial software Fluent, Inc. Gambit has a great deal of flexibility in construction of the geometry and generation of the grid and can generate unstructured multi-block meshes with hybrid elements. As both the RANS and LES solvers used in the present work have the capability to use hybrid unstructured grids, one can easily vary the mesh density so as to be able to afford a very fine mesh in the regions containing important coherent structures (e.g., meandering vortices, shed vortices, etc.). To insure maximum grid quality, which is essential for accurate LES results, we choose to avoid the use of tetrahedral elements. The unstructured grid was generated using a technique called
paving that allows the generation of unstructured meshes for surfaces and volumes that contain only rectangular and hexahedral elements, respectively.

The meshes used in both RANS (~1.5 million cells) (see Figure 3.2) and LES (close to 5 million cells) simulations (see Figure 3.3 in which the partition of the grid on 24 sub-domains, one for each processor, is also shown) are fine enough to allow integration of the governing equations up to the wall without need to use wall functions. Thus, the first row of cells off all the solid boundaries was situated at a distance such that \( y^+ \sim 0.5-2.0 \), where \( y^+ = \frac{u^+ y}{\nu} \) was estimated with the local physical Reynolds number in the hypothesis of fully turbulent flow (\( u^+/U' \sim 0.04 \), \( U' \) is the mean velocity outside the boundary layer). Some typical normal dimensions for the first row of cells near the walls are \( y=0.0005D \) \( (y^+ \sim 0.5) \) outside the pump column and \( y=0.0002D \) \( (y^+ \sim 0.5) \) for the interior pump column walls and pump bell area. At the center of the pipe the characteristic size of the cells is larger; their dimensions are around \( 30y^+ \) in all three directions. In the vicinity of the suction bell the maximum characteristic length of a cell is approximately equal to \( 6y^+ \). In the regions away from solid surfaces in the approach channel the typical size of the computational cell is around \( 30y^+ \) and the largest element size in this region is approximately \( 120y^+ \). Extensive experience with the LES solver for other massively separated flows in complex geometries suggests that this mesh density is sufficient to obtain a high quality LES solution that can accurately capture the dynamically important coherent structures in the flow.

Figure 3.4 shows a detailed view of the surface mesh on the top wall in the pump column region. A section cutting vertically through the center of the pump column is presented in Figure 3.5 where the flexibility allowed by the paving technique is evident in the very smooth transition (low stretching ratios) achieved between the near wall regions and the regions in between the intake walls and the pump column. Similar mesh quality can be observed in a section parallel to channel bottom situated at distance of \( 1.425D \) from the floor shown in Figure 3.6.

The aspect ratio of the cells, especially in the regions containing vortices, was kept as close to unity as possible including at the boundaries between the mesh blocks. Relatively low skewness of the cells was achieved in the whole domain such that a high quality LES solution can be obtained on that mesh. We estimate that by using the paving
technique we were able to decrease by a factor of 2 to 3 the total number of points required by solvers that use multi-block structured meshes if the level of mesh refinement in the critical flow regions is similar to the one achieved in the present unstructured mesh.

3.5 Discussions of Convergence and Computational Resources

In the RANS simulations at convergence the residuals of the velocity components were reduced by about 4-5 orders of magnitude from their initial values. The code was run on two processors and a fully converged steady solution was reached after about 48 hours (8,000 iterations). The convergence of the solution was checked by comparing three consecutive solutions after 8000, 10,000 and 12,000 iterations among which no noticeable changes were observed. Therefore, it was concluded that the solution was converged after 8,000 iterations. An additional RANS simulation on a grid with 1.1 million cells was conducted to check the grid independency of the solution. Comparing the results of the two simulations it was found that the position and strength of the main vortical structures in representative sections changed with less than 3%.

In the LES simulation, the total time of the simulation was 300D/U corresponding to 150,000 time steps, each of which was equal to 0.002D/U. A statistically steady state solution was obtained after 60D/U, but we started collecting statistics after 150D/U. By running the code on 24 processors we obtained around 5000 time steps per 24h on a XEON PC cluster with Myrinet. At each physical time step we iterated until the residual of u, v and w got smaller than 10^{-6} and the pressure residual got smaller than 10^{-9} corresponding to a maximum mass divergence in a cell equal to 0.005. All the threshold values were non-dimensionalized with D and U.
Figure 3.1 General view of the pump intake geometry considered in the present study
Figure 3.2 Grid used in Reynolds averaged Navier-Stoke simulations

Figure 3.3 Grid used in Large Eddy Simulations
Figure 3.4 Detailed view of the grid of the top wall of the geometry used in LES

Figure 3.5 LES grid of the present study a) view of the grid at a section through the center of the pipe; b) detail of the grid near the wall of the pump column
Figure 3.6 View of the grid at a section parallel to the channel bottom cutting through the pump column at a distance of 1.425D from the floor of the pump intake
CHAPTER 4. DISCUSSION OF RESULTS

4.1 Validation: Comparison of Results from Experiment, RANS and LES

In this section we focus on assessing the capabilities of the present LES solver to predict the mean flow and statistics in a complex pump bay geometry. Validation is performed using the results from the experiments performed by Yulin et al. (2000) who measured velocity components and turbulent kinetic energy (2 components) at various sections of the model pump intake. The statistics from LES are compared both with the experimental data and with results from a steady RANS simulation using the SST model. Specifically, the in-plane velocity components, their turbulence fluctuations and the inferred mean vorticity field are available from the experimental data at several sections parallel to the channel bottom, lateral walls and back wall of the domain. This data are used in the validation. In the LES simulation statistics were collected over 150D/U.

In all the figures where comparisons among methods are made, the experimental PIV data are restricted to the region shown as a rectangular frame on both RANS and LES plots. The position of the different planes where the comparison is made is as follows: x2, x3, x4, x5, x6, x7 are planes parallel to the side walls; x2 and x3 are located 0.92D and 1.21D away from the side wall 1 (corresponding to inlet 1 channel side); x4 is the plane passing through the symmetry axis of the domain, through the center of the pipe; x5, x6 and x7 are located 1.21D, 0.92D and 0.77D away from the side wall 2; z1, z2, z3 and z5 are planes parallel to the channel floor, located 0.4D, 0.62D, 0.9D and 1.42D away from it; y1, y3 and y4 are planes parallel to the back wall; y1 is situated 0.4D from the back wall while y3 is passing through the center of the pipe and y4 is situated 0.77D from the center of the pump column measured toward the dividing wall (Figure 4.1).

Figure 4.2 to Figure 4.9 compare the 2D mean streamlines in several representative planes situated in sub-regions of the domain, where relatively strong mean coherent vortices are present. Both RANS and LES predict the formation of a clockwise rotating vortex parallel to the back wall at the corner between the top wall and the back wall on both sides of the pipe intake (Figure 4.2 to Figure 4.5). Unfortunately, no PIV
data is available in this region to confirm the presence of this eddy. Another vortical structure predicted by both RANS and LES is a vortex on the channel 2 side situated in between the pump column and the splitter wall close to the top wall as shown in Figure 4.4 and Figure 4.5. A smaller side wall attached vortex very close to the intake bottom on the channel 1 side can be observed in Figure 4.2 in both RANS and LES results as well as in the LES results in Figure 4.3. The streamlines in the experiment appear to converge toward the lower left corner of the PIV area, in very good agreement with the patterns observed in both LES and RANS which let us speculate that the probability of the presence of the side wall 1 attached vortex in the experiment very close to the bottom is high. In the RANS results, the side wall 1 vortex is not present in the x3 plane suggesting that it diffuses faster compared to LES.

Overall, a good qualitative agreement with the mean 2D streamlines inferred from the experimental data in planes x2, x3, x4 and x5 is observed for both LES and RANS simulations. This is expected to happen as in these sections the flow near the pump bell is determined greatly by the suction of fluid from all directions toward the pipe intake. Both RANS and LES correctly capture the position of what appears to be a partial cut through the core of the main floor attached vortex which enters the pipe at y/D=-0.4 (see Figure 4.4). However, for section x7 situated closest to side wall 2 (Figure 4.6) LES captures more accurately the position and structure of the main side wall attached vortex on the channel 2 side. RANS also predicts the formation of such a vortex, however its core appears to be situated at y/D=1.5 as opposed to y/D~0.7-0.8, in both experiments and LES. Moreover, its shape is much more elongated in the direction parallel to the bottom. A secondary mean side wall vortex can be detected in the LES simulation upstream of the main eddy. This may suggest that in reality the flow in this region is driven by the interaction between the main side wall vortex and smaller secondary eddies shed from the separating boundary layer on the bottom which merge or interact quasi periodically with the main eddy. This secondary mean eddy is not present in the RANS results. As a consequence of the asymmetry in the flow caused by the larger discharge in channel 1 compared to channel 2, the side wall 1 attached vortex is smaller and weaker than the main side wall 2 vortex.
Figure 4.7 to Figure 4.9 help visualize the vortical structures in planes parallel to the bottom situated between the bottom and the pump bell level (Figure 4.7) or cutting through the pump column (Figure 4.8 and Figure 4.9). Both LES and RANS capture the formation of a very strong floor attached vortex in z1 plane (Figure 4.7). This is the main wall attached vortex in this flow and its prediction from a quantitative standpoint will be analyzed in details in a following section. For the present pump bay configuration the intensity of the floor attached vortex is determined in great measure by the imbalance between the discharges in the two channels. The position and size of the core of this vortex in Figure 4.7 appear to be very similar to the ones shown in the experiment. As one moves at higher levels, the overall vortical structure outside the pump column becomes more complex, in the sense that a vortex parallel to the vertical axis is present in planes z3 and z5 in both RANS and LES plots (Figure 4.8 and Figure 4.9). This vortex is also predicted by the experiment. A careful inspection of the plots shows that the shape and position of this predominantly vertical vortex are better reproduced in LES compared to RANS. Unfortunately, the PIV measurement area contains only the part of this eddy close to the pump column. Several mean secondary eddies are observed in the simulations immediately downstream the divider wall. As we shall see in the next section, the tip of the divider wall is in fact shedding small eddies toward the pump column and the structures observed in the mean field visualizations are a consequence of averaging over these unsteady motions.

The above qualitative comparison of the mean flow features observed in simulations and experiment is not sufficient to draw clear conclusions on the superiority of one method (turbulence model) over the other. In the following discussion, quantitative comparisons of in-plane mean velocity magnitude, turbulent kinetic energy (t.k.e.) and out-of-plane mean vorticity are used to better assess the relative performance of the LES and RANS. In the experiment the total t.k.e. is estimated using only the two in-plane instantaneous velocity components available, as ¾ of the sum of the two in-plane root mean square (r.m.s.) fluctuations. The velocity magnitude and out-of-plane vorticity field are calculated from the mean in-plane velocity components in both experiment and in simulations (statistics fields for LES).
The velocity magnitude contours in planes x6 and x7 situated toward side wall 2 are relatively well predicted by both RANS and LES (Figure 4.11 and Figure 4.12). The contour corresponding to $U_{mag}/U=0.2$ in plane x7 appears to be better reproduced in LES, as the overall shape and the size of surface encompassed by this contour situated on the right of the pipe centerline are closer to the experimental measurements. Differences are much more evident in the comparison of the distribution of the t.k.e. in the same plane shown in Figure 4.13, where only LES is able to predict the patch of relatively high t.k.e. values just beneath the pump bell level. Still the LES predictions are about 50% less than the experimental values inside that patch. This flow feature is totally absent in the RANS results. Comparison of velocity magnitude in plane y1 situated in between the back wall and the pipe shows that RANS and LES successfully predict the overall distribution of the velocity, however RANS appears to do a slightly better job than LES in capturing the shape of the isovelocity contour lines inside the central patch of relative high velocity magnitude centered just below the pump bell level (Figure 4.14). Results are also available for the velocity magnitude distribution in a plane parallel to the back wall situated at 0.77D from the pipe intake axis toward the divider wall. LES predictions are very close to experiment while RANS underpredicts by about 40% the velocity magnitude in the lower part of the PIV area where high velocity magnitudes were measured. Comparison of t.k.e. distribution in plane y1 (Figure 4.15) shows that RANS fails again to correctly predict this important turbulence quantity. The patch of very high t.k.e. levels situated in between the bottom and the pump bell level is not present in the experiment and LES. On the contrary, LES is able to accurately capture the t.k.e. distribution in this plane, including the low t.k.e. area in the upper left corner of the experimental window and the elongated high t.k.e. area on the right of the same window. The t.k.e. levels in experiment and LES are close over the whole region where PIV data is available. Furthermore, the examination of Figure 4.16 and Figure 4.17 showing a plane parallel to the back wall passing through the center of the pipe, and thus also cutting through the core of the bottom attached vortex provides maybe the best argument in favor of the necessity of using LES to accurately capture the structure and intensity of the main vortices present in pump bay configurations. The velocity magnitude distribution is clearly better predicted in LES in the area corresponding to the core of the
floor attached vortex, just beneath the pump bell level (due to limitations of PIV there are no measurements inside the pump column) and equally important RANS fails totally to capture the high t.k.e. levels inside the core of the turbulent vortex which is quite surprising. On the other hand, LES is able to capture the sharp increase of t.k.e. within the core not only qualitatively but also quantitatively. Based on the good agreement observed below the pump bell level one can suppose that LES will have a much better chance than RANS to correctly predict the t.k.e. distribution inside the pipe.

Comparison of t.k.e. within the pump column between RANS and LES shows a very different distribution of t.k.e. In LES the area of high t.k.e. appears to be situated near the center of the core of the floor attached vortex, while in the same area, RANS incorrectly predicts very low values of the t.k.e. This is quite surprising because 2D streamlines/patterns in planes parallel to the floor cutting at different levels through the pipe are quite similar in LES and RANS. Rather in RANS two patches of high t.k.e. are observed between the pipe walls and the centerline (this corresponds only to one circular region as the plane cuts through an axi-symmetric pipe). The t.k.e. levels in plane x4 (Figure 4.10) which cuts through the center of the pump column and is perpendicular to plane y3 fully confirms the above discussion of the capabilities of RANS and LES to predict the distribution of the t.k.e. within the floor attached vortex.

The velocity magnitude, t.k.e. and the absolute value of the mean out-of-plane vorticity contours in planes z1 and z2 parallel to the channel bottom are shown in Figure 4.19 through Figure 4.24. The z1 plane is situated roughly at mid-distance between the channel bottom and the pump bell level, whereas z2 plane is very close to the pump bell level. In both planes the only important vortical structure present is the floor attached vortex. As already discussed in Figure 4.7, the 2D streamline patterns are very similar in experiment, RANS and LES. A very similar observation can be made about the velocity magnitude contours in Figure 4.19. The only noticeable difference between experiment and LES on one side and RANS on the other is the very low in plane velocity magnitude value recorded at the center of the vortex in experiment and LES, which is not observed in RANS. At the pump bell level (Figure 4.22) LES better agrees with experiment compared to RANS. The patch of relatively high velocity magnitude ($U_{\text{mag}}/U>0.9$) has an annular shape roughly between $r/D=0.05$ and 0.25. At that position, in RANS, no
relative maximum for this quantity is observed. At this level both LES and RANS capture the relative velocity minimum in the center of the core. The vorticity magnitude contours shown in Figure 4.21 and Figure 4.24 for the two planes are quite similar, and no clear conclusion can be made on the overall performance of LES versus RANS for this quantity.

Examination of t.k.e. levels in Figure 4.20 and Figure 4.23 shows once again the failure of RANS to capture the expected high t.k.e. values inside the core of the floor attached vortex. For instance, the t.k.e. value at the core center in z1 plane predicted by LES is $0.260U^2$. The measured value is $0.311U^2$ while RANS predicts $0.015U^2$. The rather relatively high values of t.k.e. recorded around the core of the vortex are unphysical. In conclusion, LES successfully capture the correct behavior of t.k.e. in these sections though the decay of t.k.e. from the center of the vortex is sharper than the one observed in experiment.

Eventhough PIV data are not available, it is interesting to compare the in-plane vorticity contours in plane z5 situated 1.42D from the channel bottom (Figure 4.25). As expected high vorticity values are recorded inside the core of the vertical vortex on channel 2 side (see Figure 4.9) and inside the attached boundary layers. However, in LES larger values of the vorticity are observed in the region between the pump column and the left corner of the intake. This is a region in which in reality the flow is very unsteady due mainly to the separation of the flow around the pipe and subsequent shedding of vortices behind the pipe intake and their interaction with the main vertical vortex. The mean effects of the presence of these unsteady vortices and massive separation in this region are not captured correctly by the steady RANS model. LES does not face these limitations, and the LES statistics correctly average over instantaneous velocity fields that contain those shedding motions. This explains the larger vorticity values observed in this region which is characterized by strong vertical interactions.

4.2 3D Visualizations of the Main Vortical Structures

In this section the main coherent structures present in both the mean flow and in one of the instantaneous LES flow fields are educed using 3D streamlines.
As observed in Figure 4.26, the very coherent mean floor attached vortex is drawn inside the pump column where it eventually starts diffusing as it advances into the pipe (observe the fact that the streamlines are slowly diverging in the flow direction). Figure 4.27 tries to visualize the same vortex using the instantaneous velocity fields. Because of the strong coherence of this vortex and because its core does not meander too much (see discussion in one of the following sections) the visualizations in the two pictures are not very different. Of course, because in the visualization obtained using the instantaneous velocity fields smaller scales are present (e.g., see Figure 4.40 in which the in-plane vorticity in the pump bell level plane was plotted), and the circulatory motions inside the pipe intake are such that they make the streamlines to diverge faster from the central regions (the streamlines are launched near the center of the pump column at the pump bell level).

As is shown in section 4.3, the side wall 2 attached vortex has a very variable structure and strength in time, meanders more than the floor attached vortex and is harder to visualize using the instantaneous flow fields. However, the vortex is present at practically all times as shown in Figure 4.35 (it is not an intermittent vortical structure) and in the mean a clear vortical structure is observed at the expected location (Figure 4.28). The vortex originates at the side wall upstream of the pump column and is eventually drawn into it. It entrains fluid coming mostly from upstream, parallel to the side wall and close to the bottom. When using instantaneous velocity fields, the streamlines are observed to suggest the presence of a vortex in that region, however it is relatively hard to clearly visualize that vortex. It is interesting to discuss the case of the weaker vortex that originated on the side wall 1 side. Though this vortex is present in the mean fields, it is not always present in the instantaneous fields (the vortex is intermittent) and its instantaneous structure is most of the time very different than the one obtained through averaging. This tells us that trying to understand the dynamics of the main coherent structures in the flow based only on the mean fields can be sometimes misleading.

Figure 4.30 and Figure 4.31 visualize the vertical vortex originating at the top surface on the side wall 2 side (see also Figure 4.9). This vortex is relatively strong and present at all times (see Figure 4.37) thus its core axes, as visualized by the mean and
instantaneous flow fields, are not very different. Still Figure 4.31 suggests the fact that the side wall 2 vortex can predominantly draw fluid from different regions at different times which is consistent with the wide changes in its structure observed in Figure 4.37.

Finally, in Figures 4.32 and 4.33 the relatively weak but stable vortex (see also Figure 4.34 where the vortex is denoted as V1) which forms near the junction between the back wall and the top wall predominantly on the channel 1 side is visualized. This vortex is also visible in planes x2 to x5 showing 2D streamlines but not in x7. As it approaches the side wall 2, its core starts bending before being entrained first toward the pump column and then inside it.

4.3 Analysis of Instantaneous Flow Fields

Investigation of the dynamics of coherent structures in the pump sump region, in particular of the evolution of different wall-attached vortices will be discussed in this section. Figure 4.34, Figure 4.35, Figure 4.36 and Figure 4.37 use instantaneous 2D streamlines to visualize the evolution in time of the vortical structure of the flow in the pump sump region. The flow is characterized by the presence of several relatively stable coherent vortices that in many cases change their shape, structure, position and strength considerably as well as of less coherent structures that appear and disappear mostly due to the interactions among the larger, more stable coherent structures.

Figure 4.34, shows the evolution of the vortical structures in plane x4 that cuts through the center of the pump (frames a to i) along with the ones deduced from the mean flow fields (frame j). The 2D streamlines show a coherent vortex (V1) at the corner between the top wall and back wall. Its size and shape change very little in time, however the meandering motion of the core of this vortex is noticeable throughout frames a to i. A second eddy (V2) is observed in all frames between the divider wall and the upstream part of the pump column near the top wall. This vortex is smaller and weaker than V1 and can contain only one eddy (frames a and b) or two eddies that interact within the core (frames c to i). The second case appears to be the preferential structure, as the mean streamlines capture a vortex V2 containing two distinct eddies inside its core (frame j). The instantaneous out-of-plane vorticity contours in the same plane (Figure
4.38) shows the presence of a wide array of scales inside the pump column which interact with the floor attached vortex that is drawn into it.

The streamlines in a plane situated at 0.92D from side wall 2 are shown in Figure 4.35. The dominant vortical structure in this plane is the side wall 2 attached vortex V5 that is present in all frames. Although the position of its axis is relatively stable, its size and intensity vary considerably. This vortex interacts with a vortex attached to the top wall which has an axis parallel to the pump column (V12 in Figure 4.37) until it starts feeding fluid into V5. A secondary side wall attached vortex (V6) situated very close to the bottom is present practically in all frames. The circulation inside this vortex has the same sign as the one inside V5. The two vortices interact among them in a strong non-linear way and with a counter-rotating bottom attached vortex present in between them. The bottom attached vortex merges at random times with V5 (see also Figure 4.39 in which the instantaneous out-of-plane vorticity contours are plotted). In fact the vorticity plots in Figure 4.39 confirm that V5 is the strongest vortical structure in this plane at all times. These interactions produce large variations in the structure of V5 in time. Smaller vortices close to the top and bottom back wall corners are also observed, however they tend to be very intermittent. For instance V3 is present in frames a, c and d while V7 is present in frames a, c, d, h and i. Their shapes and strengths are highly variable. In frames g and h, a large structure is observed near the back wall in the upper part of the domain. This structure appears to feed high momentum fluid into the core of the vortex originating at the top wall (V12). Finally, a very intermittent structure (V4) is observed in some of the frames (b, c and i). One suspects its presence is a consequence of the interactions between V5 and V12. The streamline pattern shown in the mean solution contains vortices V5, V6, V7 and V12 as well as the floor attached vortex between V5 and V6.

Figure 4.36 shows the vortical structures in plane z2 situated very close to the pump bell level. As inferred also from the out-of-plane vorticity contours in Figure 4.40 the most intense vortex in this plane is the floor attached vortex, V8. However, if in frame j that shows the mean flow, the patch of vorticity associated with V8 is clearly delimited from the surrounding low vorticity flow, in the instantaneous frames a wide range of eddies is present around the patch of vorticity corresponding to the main floor.
attached vortex. If the streamlines suggest that its shape and position appear to be fairly stable in time, the vorticity contours show that the shape of the main patch of vorticity associated with V8 can vary considerably in time. Further quantitative analysis of this vortex proved that there are relatively important variations of its intensity and core size in time (Figure 4.43 and Figure 4.44). Secondary intermittent vortices are present near the corner between the back wall and side wall 1 (e.g., frames d to h) and near side wall 2 where a region containing typically several small vortices (V10) is observed in most of the frames. Sometimes (e.g., see frame f) these vortices appear to merge into a stronger one. The other intermittent coherent structure is V9 (see also Figure 4.39). In frames f, g and h, only one vortex is present while in other frames a double vortex structure is observed (frames a and i). It is not clear if V9 is connected to the vortex originating from the top wall (V12).

At a higher level (z/D=1.13) from the floor (Figure 4.37) the main coherent structure originates at the top wall (V12). As opposed to the other main wall attached vortices V5 and V8, its structure is found to vary greatly over time. In fact the out-of-plane instantaneous vorticity contours in Figure 4.41 show several small patches of vorticity associated with the eddies inside this structure but most of the time no main patch of vorticity associated with this vortex. The mean vorticity fields (frame j) do show the presence of a patch of high (negative) vorticity corresponding to V12. Only one eddy is present in frames a to c. A secondary vortical structure V13 is present in frame a and appears to merge with V12 immediately. The structure of V12 contains two main eddies of varying strength over most of the next 90D/U. Another interesting phenomenon captured by the LES simulation (Figure 4.41) is the shedding of vortices in the form of fairly vertical vortex tubes of opposite signs from the tip of the divider wall. This vorticity is convected toward the pump column and interacts with the attached boundary layers on the exterior of the pipe or directly with the floor attached vortex at lower levels. Another area where unsteady shedding is present is behind the pump column. This shedding is similar to the one that is observed for the flow past long cylinders in a sheared incoming flow (the shear in our case is due to the mean velocity unbalance between the two channels). This makes the wake and associated vortices to tilt toward side wall 2. Because of the presence of the back wall and of V12, these shed vortices
(e.g. V14 in frame g, Figure 4.37) start interacting with V12. Frame h of the same figure shows such a scenario, when the shed vortex just before it merges with one of the main eddies inside V12. This results into an increase of the size of V12 which occupies most of the area between the pipe and the corner of side wall 2.

Figure 4.42, Figure 4.43 and Figure 4.44 try to estimate in a more quantitative way the changes in the position of the floor attached vortex, the variation of its core size and circulation in time. The analysis is done in a plane situated at 0.2D from the floor. Given a small threshold value for the vorticity, one can define the core of the vortex, calculate its area and circulation and then define a mean core radius \( r = \sqrt{A / \pi} \). The center of the mean vortex \((x_0, y_0) = (0.05D, -0.05D)\) obtained from the mean flow fields is shown in Figure 4.42. This mean value was obtained by averaging over 150D/U nondimensional time units. The 140 points in Figure 4.42 are spaced 0.5D/U apart, from 45D/U to 115D/U (origin corresponds to the start of the statistically steady interval). Interestingly, the axis of the vortex appears to oscillate within two preferential regions, one very close to the long time average position for t>60D/U and one around \((0, -0.17D)\) for 45D/U<t<60D/U. During this second time interval the radius and circulation of the vortex are larger by about 15-25% (Figure 4.43 and Figure 4.44, respectively) so one can associate this second region with transient states when the vortex is stronger and larger probably due to merging with a patch of vorticity of same sign.

The meandering of the main side wall 2 attached vortex is investigated in Figure 4.45. Again \((x_0, y_0)\) corresponds to the center of the vortex calculated from the LES mean flow fields. Larger variations of the axis of the side wall vortex, especially in the normal direction \((-0.4<z/D<-0.1\) for 45D/U<t<60D/U\), are observed compared to the floor attached vortex, due to reduced coherence and the more pronounced meandering of the core. The movement of the core over the period investigated appears to be chaotic and can be inferred from Figure 4.45.
4.4 Time Series and Power Spectra of Instantaneous Quantities

Time series of the three velocity components and pressure at sixty stations located at several levels below and around the pump column were recorded over 150D/U (flow was statistically steady) with a time step of 0.002D/U. From these time series the power spectra of the velocity and pressure were calculated. Out of the sixty stations, results at five representative points are presented.

Pressure power spectra and time series inside the floor attached vortex are shown in Figure 4.46 for two stations that are situated at 0.2D (Figure 4.46a) and 0.6D (Figure 4.46b), at pump bell level, from the channel bottom. The sudden depressions observed in the pressure time series are associated with the meandering of the core of the floor attached vortex. As inside the core of a vortex the pressure is smaller, the presence of a depression means that the station is situated very close to the axis of the vortex at that moment. The main frequency in the region between the pump bell and the floor corresponds to a Strouhal number \( \text{St} = \frac{fD}{U} = 0.14 \) (~1.74Hz). Secondary peaks corresponding to the first (\( \text{St}=0.28 \)) and third harmonics (\( \text{St}=0.56 \)) of the fundamental frequency are observed. The presence of these harmonics suggests that strong non-linear interactions are present inside the floor attached vortex as well as between this vortex and the surrounding turbulence (see also Figure 4.40). The spectrum is broad, as expected inside a region where the flow is highly turbulent. Examination of the time series show that the small scale resolved (high frequency) oscillations of the pressure signal are produced in a frequency band corresponding to \( \text{St} \sim 2.6-2.9 \) at both locations. These frequencies are also visible in the plots showing the spectra for \( \text{St}<20 \).

Figure 4.47 shows time series and power spectra at a station located inside the pump column \((z/D=0.9)\) which helps us to track the changes in the spectral characteristics of the floor attached vortex as it gets entrained into the pipe. The time series of the velocity suggest that the flow inside the upstream part of the pump column is highly turbulent (Figure 4.47a) due to the entrainment of vorticity from the different wall attached vortices and separation taking place at the mouth of the pump bell. Sudden reductions in the pressure are again noticeable in Figure 4.47b corresponding to the meandering of the core of the floor attached vortex inside the pump column. As expected
the dominant frequency inside the pump column is the same as the one recorded inside the floor attached vortex below the pump bell, however a second energetic frequency is present at St=0.21 along with its first harmonic (St=0.42). However, the most important feature of the velocity and pressure spectra inside the pump column is the presence of a band of high energetic frequencies around St=14.5 (~18Hz). This is visible especially in the log-log scale power spectra in Figure 4.47 where the energy increases by two decades around St=14.5. These frequencies are associated with the lowest period oscillations in the pressure/velocity time series and they are well resolved in our simulations (~30 time steps).

Figure 4.48 and Figure 4.49 display the pressure time series and spectra at a point situated inside the side wall attached vortex (V5 in Figure 4.35) in a plane located at 0.54D from side wall 2 and at a station situated inside the corner vortex (V1 in Figure 4.34) near the junction between the back wall and the top intake wall, respectively. For the side wall attached vortex, two energetic frequencies associated with the oscillations/meandering of the core are observed at St=0.22 and St=0.28 along with their harmonics. For the corner vortex the main energetic frequencies are present at St=0.19 and St=0.23. Though a band of relatively energetic frequencies appears to be present around St=14.5 at both stations, the energy increase relative to the surrounding frequencies is much smaller compared to the one observed at the station located inside the pump column.

4.5 Decay of Vorticity inside the Pump Column

To improve the performance of the pumps, it is desirable to achieve low swirl flow conditions inside the pump column at the pump level. The level of swirl inside the pump column is dependent on the overall strength of the circulatory motions inside the pump bay, on the intensity of the various subsurface vortices and on that of the separation eddies shed around the pump bell. In the case studied in the present work, the most important factor is the intensity of the floor attached vortex that is injected inside the pump column. The decay of the swirl inside the pipe intake was computed from the LES statistics. Figure 4.50 shows the out of plane mean vorticity contours at different levels inside the pipe starting at the pump bell level. The patch of vorticity associated with the
core of the floor attached vortex is visible in all these sections. As shown in Figure 4.51 in which the circulation associated with this patch of vorticity was calculated, the intensity of the swirl is found to decay by 90% between the pump bell and a level situated at 1.7D from it, mainly due to turbulent dissipation effects. Then, the decay is much milder and appears to have a constant slope. The CFD model can be used to estimate what would be the minimum length of the pump column upstream of the pumps needed such that the level of circulation at the pump will not be detrimental to its efficiency.

Another major advantage of an LES simulation is that it provides continuous swirl (circulation) data at any level inside the pump column which can be used to study the time evolution (e.g., variance) of the swirl. Sometimes the circulation is converted into a swirl angle and then the quantities of interest are the average (obtained from the statistics) and the maximum swirl angle (obtained from time series corresponding to the instantaneous flow fields). This kind of information is very important, as in many cases the performance of the pumps is affected not only by the mean value of the swirl (circulation) which can also be obtained from RANS (though the accuracy of the prediction will be most probable lower) but also by other unsteady characteristics of the swirl variation that are not available from a steady RANS simulation. For instance, as pointed out by Johansson et al. (2005), the intermittent and direction changing swirl inside the pump column may be more harmful to the pump efficiency than a steady one-directional swirl of similar magnitude. In other cases the mean circulation value can be very low; however the instantaneous values can vary significantly about the mean assuming large positive and negative values. Deciding the pump position based only on the mean value of the swirl in such cases will certainly be inappropriate.

For instance, Figure 4.52 shows the time variation of the total circulation (the vorticity contribution from the attached boundary layers on the pump column walls was eliminated) inside the pump column in a section situated at 0.6D from the pump bell level, along with the positive and negative components to the total swirl. The horizontal line corresponds to the long time average in the same section. Frame b shows the mean vorticity circulation at the same section from LES statistics while frame c shows one of the instantaneous vorticity fields used to produce the circulation time series. Figure 4.53 displays the same information but for a plane situated downstream in the pump column at
2.0D from the pump bell level. What is interesting to remark is not only the fact that the total circulation displays large variations around the mean values (in this sense a simpler RANS model should ideally have the capability to predict at the minimum some measure of the variance of the swirl along with the mean value) but also that it changes sign pretty regularly not only close to the pump bell level where due to the massive separation and injection of different vortices this is somewhat expected but also at sensible higher levels inside the pump column. For example, in Figure 4.53 the change in swirl direction is found to occur six times over 140D/U with the maximum absolute values of the circulation occurring around 40D/U and 115D/U during these reversal periods. In the same figures, the positive component of the total circulation which is associated with the injection of the floor attached vortex inside the pump column was plotted. This is the only vortical structure that maintains its coherence over long distances inside the pump column. By comparing the variation of its circulation (positive component) with the one of the total circulation, one can clearly see that the large variations in the total circulation are highly correlated with variations in the intensity of the floor attached vortex, with the negative component oscillations being sensible smaller compared to the ones observed in the positive component. This proves that, for the geometry and flow simulated, reducing the intensity of the floor attached vortex is essential to reduce not only the mean level but also the amplitude of the swirl oscillations at the pump level inside the pump column.
Figure 4.1 Sections where experimental measurements, RANS and LES results are compared
Figure 4.2 2D streamlines from experiment, RANS and LES in x2 plane situated at 0.92D away from side wall 1

Figure 4.3 2D streamlines from experiment, RANS and LES in x3 plane situated at 1.21D away from side wall 1
Figure 4.4 2D streamlines from experiment, RANS and LES in x4 plane cutting through the center of the pump column

Figure 4.5 2D streamlines from experiment, RANS and LES in x5 plane situated at 1.21D away from side wall 2
Figure 4.6 2D streamlines from experiment, RANS and LES in x7 plane situated at 0.77D away from side wall 2

Figure 4.7 2D streamlines from experiment, RANS and LES in z1 plane situated at 0.4D away from channel bottom
Figure 4.8 2D streamlines from experiment, RANS and LES in z3 plane situated at 0.9D away from channel bottom

Figure 4.9 2D streamlines from experiment, RANS and LES in z5 plane situated at 1.425D away from channel bottom
Figure 4.10 Turbulent kinetic energy contours in x4 plane cutting through the center of the pump column

Figure 4.11 Velocity magnitude contours in x6 plane situated at 0.92D away from side wall 2
Figure 4.12 Velocity magnitude contours in x7 plane situated at 0.77D away from side wall 2

Figure 4.13 Turbulent kinetic energy contours in x7 plane situated at 0.77D away from side wall 2
Figure 4.14 Velocity magnitude contours y1 plane situated at 0.4D away from the back wall

Figure 4.15 Turbulent kinetic energy contours y1 plane situated at 0.4D away from the back wall
Figure 4.16 Velocity magnitude contours in y3 plane cutting through the center of the pump column

Figure 4.17 Turbulent kinetic energy contours in y3 plane cutting through the center of the pump column
Figure 4.18 Velocity magnitude contours in y4 plane situated at 0.77D away from the center of the pump column parallel to the back wall

Figure 4.19 Velocity magnitude contours in z1 plane situated at 0.4D away from channel bottom
Figure 4.20 Turbulent kinetic energy contours in z1 plane situated at 0.4D away from channel bottom

Figure 4.21 Out-of-plane absolute vorticity contours in z1 plane situated at 0.4D away from channel bottom
Figure 4.22 Velocity magnitude contours in z2 plane situated at 0.6D away from channel bottom

Figure 4.23 Turbulent kinetic energy contours in z2 plane situated at 0.6D away from channel bottom
Figure 4.24 Out-of-plane absolute vorticity contours in $z_2$ plane situated at 0.6D away from channel bottom

Figure 4.25 Out-of-plane vorticity contours in $z_5$ plane situated at 1.425D away from channel bottom
Figure 4.26 Visualization of floor attached vortex; using 3D streamlines from mean LES solution (statistics)

Figure 4.27 Visualization of floor attached vortex using 3D streamlines from an instantaneous LES solution
Figure 4.28 Visualization of side wall attached vortex using 3D streamlines from mean LES solution (statistics)

Figure 4.29 Visualization of side wall attached vortex using 3D streamlines from an instantaneous LES solution
Figure 4.30 Visualization of the vertical vortex between pump column and side wall using 3D streamlines from mean LES solution (statistics)

Figure 4.31 Visualization of the vertical vortex between pump column and side wall using 3D streamlines from an instantaneous LES solution
Figure 4.32 Visualization of the vortex between back wall and pump column using 3D streamlines from mean LES solution (statistics)

Figure 4.33 Visualization of the vortex between back wall and pump column using 3D streamlines from an instantaneous LES solution
Figure 4.34 Instantaneous streamlines in the symmetry plane $x_4$ cutting through the center of the pipe and parallel to the side walls; a) $t = 0$; b) $t = 15D/U_0$; c) $t = 30D/U_0$; d) $t = 45D/U_0$; e) $t = 60D/U_0$; f) $t = 75D/U_0$; g) $t = 90D/U_0$; h) $t = 105D/U_0$; i) $t = 120D/U_0$; j) mean streamlines from averaged LES solution
Figure 4.35 Instantaneous streamlines in x6 plane parallel to the side wall situated at 0.92D from side wall 2; a) t = 0; b) t = 15D/U₀; c) t = 30D/U₀; d) t = 45D/U₀; e) t = 60D/U₀; f) t = 75D/U₀; g) t = 90D/U₀; h) t = 105D/U₀; i) t = 120D/U₀; j) mean streamlines from averaged LES solution
Figure 4.36 Instantaneous streamlines in z2 plane parallel to channel floor at the pump bell level (0.6D from the floor); a) $t = 0$; b) $t = 15D/U_0$; c) $t = 30D/U_0$; d) $t = 45D/U_0$; e) $t = 60D/U_0$; f) $t = 75D/U_0$; g) $t = 90D/U_0$; h) $t = 105D/U_0$; i) $t = 120D/U_0$; j) mean streamlines from averaged LES solution
Figure 4.37 Instantaneous streamlines in z4 plane parallel to the channel bottom situated 1.13D from channel floor; a) $t = 0$; b) $t = 15D/U_0$; c) $t = 30D/U_0$; d) $t = 45D/U_0$; e) $t = 60D/U_0$; f) $t = 75D/U_0$; g) $t = 90D/U_0$; h) $t = 105D/U_0$; i) $t = 120D/U_0$; j) mean streamlines from averaged LES solution
Figure 4.38 Instantaneous out-of-plane vorticity contours in the symmetry plane x4 cutting through the center of the pipe and parallel to the side walls; a) $t = 0$; b) $t = 15D/U_0$; c) $t = 30D/U_0$; d) $t = 45D/U_0$; e) $t = 60D/U_0$; f) $t = 75D/U_0$; g) $t = 90D/U_0$; h) $t = 105D/U_0$; i) $t = 120D/U_0$; j) mean out-of-plane vorticity from averaged LES solution
Figure 4.39 Instantaneous out-of-plane vorticity contours in x6 plane parallel to the side wall situated at 0.92D from side wall 2; a) t = 0; b) t = 15D/U₀; c) t = 30D/U₀; d) t = 45D/U₀; e) t = 60D/U₀; f) t = 75D/U₀; g) t = 90D/U₀; h) t = 105D/U₀; i) t = 120D/U₀; j) mean out-of-plane vorticity from averaged LES solution
Figure 4.40 Instantaneous out-of-plane vorticity contours in z2 plane parallel to channel floor at the pump bell level (0.6D from the floor); a) t = 0; b) t = 15D/U₀; c) t = 30D/U₀; d) t = 45D/U₀; e) t = 60D/U₀; f) t = 75D/U₀; g) t = 90D/U₀; h) t = 105D/U₀; i) t = 120D/U₀; j) mean out-of-plane vorticity from averaged LES solution
Figure 4.41 Instantaneous out-of-plane vorticity contours in z4 plane parallel to the channel bottom situated 1.13D from channel floor; a) t = 0; b) t = 15D/U0; c) t = 30D/U0; d) t = 45D/U0; e) t = 60D/U0; f) t = 75D/U0; g) t = 90D/U0; h) t = 105D/U0; i) t = 120D/U0; j) mean out-of-plane vorticity from averaged LES solution.
Figure 4.42 Meandering of the core of the floor attached vortex in a section situated at 0.2D from the channel floor in time

Figure 4.43 Evolution of the mean core radius of the floor attached vortex in a section situated at 0.2D from the channel floor
Figure 4.44 Evolution of the absolute circulation of the floor attached vortex in a section situated at 0.2D from the channel floor.

Figure 4.45 Meandering of the core of the side wall 2 attached vortex in a section situated at 0.77D from the side wall 2.
Figure 4.46 Spectral analysis of floor attached vortex; a) pressure time series and power spectra at a point situated near the channel floor below the pump bell level; b) pressure time series and power spectra at a point situated at the pump bell level.
Figure 4.47 Spectral analysis of floor attached vortex at a point inside the pump column; a) y-velocity time series and power spectra; b) pressure time series and power spectra; c) x-velocity power spectrum in log-log scale and in linear-linear scale; d) z-velocity power spectrum in log-log scale and low and in linear-linear scale
Figure 4.48 Spectral analysis of side wall 2 attached vortex; a) time series of the pressure; b) pressure power spectrum in log-log scale; c) low frequency part of pressure spectrum; d) high frequency part of pressure spectrum
Figure 4.49 Spectral analysis at a point situated near the junction between the back wall and the upper wall; a) time series of the pressure; b) pressure power spectrum in log-log scale; c) low frequency part of pressure spectrum; d) high frequency part of pressure spectrum
Figure 4.50 Mean out-of-plane vorticity contours at different levels inside the pipe obtained from the statistics of LES; a) level $z = 0$; b) level $z = 0.6D$; c) level $z = 1.0D$; d) level $z = 1.5D$; e) level $z = 1.7D$; f) level $z = 2.0D$; g) level $z = 2.5D$; h) level $z = 3.0D$
Figure 4.51 Decay of absolute circulation inside the pump column

Figure 4.52 Circulation inside the pump column in a plane 0.6D from the pump bell level; a) evolution of circulation in time; b) mean out-of-plane vorticity contours; c) instantaneous out-of-plane vorticity contours
Figure 4.53 Circulation inside the pump column in a plane 2.0D from the pump bell level; a) evolution of circulation in time; b) mean out-of-plane vorticity contours; c) instantaneous out-of-plane vorticity contours
CHAPTER 5. CONCLUSIONS AND RECOMMENDATIONS

The goal of this study was to show that advanced CFD models using eddy resolving techniques (LES) and non-dissipative solvers can accurately predict the complex flow phenomena present in pump intake structures.

The capabilities of a novel non-dissipative LES code and of a performant commercial RANS code to predict the mean flow and vortical structure of the flow in a water pump intake of realistic geometry containing vortex suppressing devices were compared. This is the first study in which a non-dissipative LES solver and fine meshes (of the order of 5 million cells) were used to investigate the flow in a pump-bay of realistic geometry followed by in-depth validation from an experimental model study using PIV techniques.

The second part of the study concentrated on the fundamental flow physics and tried to visualize the evolution of the main coherent structures including phenomena such as intermittency and meandering, their spectral content and their interactions using the LES flow fields. In this regard the present LES simulations enabled, for the first time, a comprehensive three dimensional description of the unsteady pump bay vortices and their interactions using CFD.

The present study proved once again the need having available a set of comprehensive and high quality experimental data to fully validate an eddy resolving numerical model of such complex flows. In this regard, sometimes it was rather difficult to draw a clear conclusion on the predictive performances of the numerical models because experimental data were not available at some critical regions of the flow. Also, no visualizations of the instantaneous/mean main wall attached vortices were available. This kind of data would have been of great help especially for further validation of the LES model. However, overall the experimental data set collected by Yulin et al. (2000) provided a unique and fairly comprehensive data set that was successfully used to validate the numerical models and to compare their predictive abilities.

It is also important to emphasize that in the simulations, especially in LES, the quality of the mesh plays a critical role in the performance and robustness of the models. The use of models and codes capable of using unstructured multi-block meshes allowed
the generation of a high quality mesh over the whole domain with a smooth transition between regions where mesh refinement was necessary due to the presence of an attached boundary layer or a vortex and regions where a coarser mesh can be employed. This facilitated an important reduction (especially for LES) in the total number of mesh points necessary to accurately simulate and resolve the flow at a given Reynolds number. Compared to a structured multi block grid system we estimate that we reduced the total number of cells by at least a factor of two for the same mesh quality. Moreover, the use of unstructured topologies and hexahedral meshes obtained using paving techniques appear to allow generation of high quality meshes in very complex domains while keeping the total number of faces and nodes at a minimum given a certain characteristic cell size.

Both models avoided the approximations associated with the use of the wall function approach by considering meshes that were highly refined in the wall normal direction in the vicinity of the solid surfaces. The robustness of the LES solver for the present application in which the physical Reynolds number was relatively high was a consequence of the fact that the finite volume algorithm discretely conserves not only mass and momentum but also energy.

The present simulations showed that both RANS and LES are fairly successful in reproducing the mean flow features (e.g., location, shape, size of the main wall attached vortices) observed in the experiment. Qualitatively both methods were found to produce similar flow patterns in relevant sections with a slight advantage to LES, as the comparison of 2D streamlines suggested. A similar conclusion was reached for the distribution of the in-plane mean velocity magnitude and out-of-plane vorticity, with LES being again closer to the experiment.

On the other hand, present simulations clearly show that RANS failed to predict the turbulent kinetic energy distribution throughout the flow, in particular in the region where the floor attached vortex, was present.

The floor attached vortex was the strongest coherent structure in the flow, a consequence of the asymmetry in the approach flow conditions and was found to determine to a great measure the level of swirl inside the pump column. Of course, a steady RANS simulation cannot capture the vortex shedding or the
meandering/intermittency of the main coherent structures. In fact even a time-accurate RANS will most probably fail in predicting this type of phenomenon due to the fact that URANS models do not allow the energy cascade to take place in a physical way and are too dissipative in critical regions of the flow.

Another advantage of using LES which is inherently time accurate is that it allows observation of the evolution of vortices in the flow throughout the simulation and concomitantly over the whole computational domain something that even PIV cannot achieve. This is especially important for flows in which intermittent and/or meandering vortices are present, thus making very difficult understanding the interaction between the different vortices based on information collected over different time intervals as would be the case in a PIV investigation. On the other hand, LES does not face these limitations and enables a detailed investigation of the dynamics of the main coherent structures and of their interactions in both space and time making it a very powerful tool to investigate the physics of highly unsteady and vorticity dominated flows. Understanding the fundamental flow physics and the interaction between the vortices is a first step toward controlling them and eventually being able to reduce their intensity or eliminate them in the redesign process.

In the present study, the evolution and changes in the structure of the main vortical features originating at the floor, lateral walls and top wall including phenomena such as intermittency and meandering were analyzed using the instantaneous LES flow fields and spectral analysis. In particular, for the pump intake geometry and flow conditions studied, the core of the floor attached vortex was found to maintain its coherence not only at the pump bell level but also over the whole extent of the vertical pump column. Using spectral analysis at locations close to or inside the dominant vortices, several interesting flow features were observed. For instance, the presence of energetic frequency components corresponding to the first and third harmonics of the fundamental frequency confirmed the existence of strong non-linear interactions between the eddies inside the floor attached vortex and the surrounding flow. A band of very high but fairly energetic frequencies at about 100 times the value of the fundamental frequency was observed inside the upstream part of the pump column.
The decay of the induced swirl inside the pump column was quantified. The numerical simulation has shown that there is a region (~1.7D from the pump bell level) in which the intensity of the swirl inside the pump column decays fast after which the decay is much slower. Given a certain threshold value for the circulation at the pump levels, one can thus estimate the optimal location of the pumps and thus avoid the decay in pump performance due to adverse effects related to the entrainment of the floor attached vortex into the pump column. Analysis of the time series of the swirl downstream in the pipe intake showed that the instantaneous swirl values can oscillate considerably about the mean. Variance along with the mean value of the swirl should be considered when deciding the position of the pumps.

To sum up, the present study has shown that an accurate non-dissipative LES model that uses a sufficiently fine mesh can accurately capture not only qualitatively but also quantitatively most of the mean flow features observed in an experimental investigation using PIV of the same flow and can shed light into the complex process through which the main coherent structures interact with each other. The biggest challenge of the LES approach is to reduce the time needed to obtain a solution compared to RANS codes. Further advance in computer technology and numerical algorithms will help address this problem and will make LES a viable alternative to RANS for the numerical studies typically employed in the design process.

As the LES model uses hybrid multi-block unstructured meshes it can be used without additional changes to simulate flow in a pump intake with multiple pumps, or extended upstream to include the reach of the river with the diversion channel. The present LES based CFD model may be coupled with similar models to simulate the flow through other components of the flow system. For example, the results of this CFD model may be used to specify the inlet conditions for calculation of the flow through the pumps to determine conditions leading to cavitation or calculate the loadings on the impeller. Also LES can be used to optimize the shape and optimal positioning of vortex suppressing devices currently used in the redesign process (e.g., floor cones, back wall splitters or dividing walls). The numerical model should be further refined by improving the modeling of the free surface dynamics for cases in which the pump intakes is not pressurized. This will necessitate a more direct treatment of the free surface boundary
conditions and considering the effect of the surface tension, and ultimately, the phenomenon of free surface breaking. The numerical model may also be employed to study sediment transport, and thus be able to address the problem of sediment deposition at water intakes. Another area of future research is the simulation of pump intake flows at full scale Reynolds numbers which would allow the study of the influence of scale effects. However, since a well resolved LES is too expensive, the use of hybrid LES/RANS approaches like Detached Eddy Simulation (DES) or LES models with complex wall treatment which use RANS models in the near wall region and then transition to the classical Smagorinsky model away from solid boundaries is envisioned. As these approaches introduce additional modeling assumptions and parameters, comprehensive validation is an even more essential requirement.

The long time goal of the present work is to use LES as a predictive tool that can be employed in the design or redesign process of pump intakes thus replacing at least partially the expensive and site-specific laboratory studies that are used at the present time. As LES directly resolves the most energetic coherent structures, it has built into it much more physics compared to RANS/URANS models and thus a much better chance to accurately capture the unsteady dynamics of the vortices and to thus more accurately predict the mean flow.
REFERENCES


