Numerical assessment of the performance of a wastewater pump under off-design operating conditions

A thesis submitted to the
Department of Mechanical, Manufacturing and Biomedical Engineering
Trinity College Dublin, the University of Dublin
in partial fulfilment of the requirements for the degree of

Doctor of Philosophy

by

Federico Caruso

Supervisor:
Dr. Craig Meskell

July, 2021
Declaration of Authorship

I, Federico CARUSO, declare that this thesis titled “NUMERICAL ASSESSMENT OF THE PERFORMANCE OF A WASTEWATER PUMP UNDER OFF-DESIGN OPERATING CONDITIONS” has not been submitted as an exercise for a degree at this or any other university and it is entirely my own work.

I agree to deposit this thesis in the University’s open access institutional repository or allow the library to do so on my behalf, subject to Irish Copyright Legislation and Trinity College Library conditions of use and acknowledgement.

Signed:

__________________________________________
Alla mia famiglia
"Ad maiora semper"
Abstract

Pumps are often key elements of industrial processes, and are commonly used to move water in municipal water schemes and in wastewater treatment of industrial effluents, representing a substantial energy cost. In the wastewater industry, pumps often have a single blade impeller with non-clog centrifugal designs, as the main design criterion is to allow the passage of solid objects through the vane without blockage. Due to the asymmetric geometry of the vane, the vibrations induced by the radial thrust of the impeller may be severe, significantly shortening the life of bearings and seals. This increases leakage through the pump components, and thus affects the device performance. In addition, the implementation of more stringent energy optimization strategies to reduce the operational costs, such as demand-side management, including load shifting to off-peak periods and adoption of variable speed drives, will require industrial machinery to work at part load more often. As a result, it is likely that pumping systems will be run at off-design conditions, with significant effects on the energy consumption.

The scope of this thesis is to assess the performance of a wastewater pump at off-design conditions. A high fidelity three-dimensional computational model has been developed from a commercially available version of a single-blade wastewater pump (Sulzer XFP PE-2 150E CB 1.1). Numerical simulations are run using unsteady Reynolds-Averaged Navier-Stokes simulations with ANSYS Fluent and results are compared to experimental data and published data to provide confidence in the modelling approach. The transient behaviour of the wastewater pump is also assessed by varying the back-pressure and the rotational speed of the pump in a one-dimensional pipeline system. While it is found that the transient behaviour does not have a significant impact on the overall performance of the pump, the transition strategy between two operating conditions has a significant impact on the peak power and the total energy input. The effect of the axial gap between the impeller and the casing is investigated and non-dimensional coefficients are defined to estimate the performance drop with the gap size. It is found that the efficiency can drop up to 13.5 %·mm⁻¹. The outcomes emphasize the need to include specific maintenance protocols to ensure axial clearance is kept close to design targets.
# Contents

Declaration of Authorship  iii

Abstract  ix

1 Introduction 1
   1.1 Scope of this thesis  3
   1.2 Summary  3

2 Literature Review 5
   2.1 Demand-side management  7
   2.2 The water-energy nexus  9
   2.3 Energy costs in wastewater treatment plants  12
   2.4 Hydraulic pumps energy costs and saving opportunities  17
   2.5 Computational methods for centrifugal pumps  21
   2.6 Summary  26

3 Centrifugal pumps performance 29
   3.1 Performance characteristics  30
   3.2 Pumping systems and control  34
   3.3 Variable speed control  35
   3.4 Viscous effects  37

4 Numerical Methods and setup 39
   4.1 Turbulence modelling  39
   4.2 Near-wall modelling  41
   4.3 Solution algorithms  44
   4.4 Wastewater pump geometry  48
      4.4.1 Geometry modelling  50
      4.4.2 Meshing strategy  53
      4.4.3 Solver setup  57

5 Transient solution at constant rotational speed 61
   5.1 Internal flow results  62
   5.2 Original and modified trailing edge comparison  70
5.3 Performance metrics with pressure-inlet pressure-outlet boundary conditions . . 72
5.4 Summary . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 74

6 Unsteady force excitation 77
6.1 Unsteady forces . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 78
6.2 Spectral components . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 83
6.3 Summary . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 88

7 Effect of the axial gap on pump performance at off-design conditions 89
7.1 Effect of the gap on performance metrics . . . . . . . . . . . . . . . . . . . . . . . 89
   7.1.1 Sensitivity of performance to gap size . . . . . . . . . . . . . . . . . . . . . 93
7.2 Assessment of the flow velocity through the gap . . . . . . . . . . . . . . . . . . . 97
7.3 Summary . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 100

8 Transient performance with variable speed drives at off-design conditions 103
8.1 Pressure control . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 103
8.2 Combined pressure-speed control . . . . . . . . . . . . . . . . . . . . . . . . . . . 108
8.3 Transient behaviour with the gap size . . . . . . . . . . . . . . . . . . . . . . . . . 113
8.4 Scaling parameters . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 115
8.5 Summary . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 117

9 Conclusions 119
9.1 Future developments . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 121

A UDF codes 123
   A.1 Static pressure UDF . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 123
   A.2 Rotational speed UDF . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 124

B Computational methods 125
   B.1 Governing equations . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 128
       B.1.1 Conservation of mass . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 129
       B.1.2 Conservation of momentum . . . . . . . . . . . . . . . . . . . . . . . . . . . 129
       B.1.3 Conservation of energy . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 130

C Publications 133
   C.1 Journal Articles . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 133

D Datasheet PE90/4-50Hz XFP150ECB1 135
## List of Figures

1.1 Exploded view of the single-blade pump model: wear plate, impeller, volute casing. 2

2.1 World energy data (2018). ................................................................. 5

2.2 Load levelling from peak-load shifting [Yang and Jackson, 2011]. .............. 7

2.3 Hybrid Sankey diagram of the U.S. water-energy nexus in 2011 [Greenberg et al., 2017]. ................................................................. 11

2.4 The water-energy nexus in WWTP [Feng and Chen, 2016]. ......................... 12

2.5 Simplified diagram of the main stages of wastewater treatment plant [Jazbec and Turner, 2020]. ................................................................. 13

2.6 Wastewater pumping is typically 12% of the energy consumption or approximately 0.05 kWh/m$^3$. ................................................................. 14

2.7 Pumping system with pump drive train [Arun Shankar et al., 2016]. .......... 18

2.8 Operating points and efficiencies with varying speeds [Marchi et al., 2012]. .... 20

3.1 Submersible sewage pump cross sections, Flygt [Flygt, 2015, Modopump, 2020]. 29

3.2 Ranges of specific speeds for typical pump geometries [Brennen, 2011]. ......... 32

3.3 Typical pump characteristic curves. .................................................... 33

3.4 Characteristic curves intersection determines the operating point [Al-Khalifah and McMillan, 2012]. ................................................................. 34

3.5 Variable speed drive system diagram and torque control [ABB, 2011]. ......... 36

3.6 Viscous effects within a centrifugal pump [Brennen, 2011]. ....................... 37

4.1 Velocity profile as function of the distance normal to the wall [Fluent, 2017]. ... 42

4.2 Segregated (left) and coupled (right) pressure-based algorithms [Fluent, 2017]. 45

4.3 The PISO scheme [Fluent, 2017]. ......................................................... 47

4.4 Exploded view of the single-blade pump model: wear plate, impeller, volute casing. 48

4.5 Sulzer XFP PE2 150E CB 1.1 experimental data set. ................................ 49

4.6 Original (left) and modified (right) volute. ........................................... 50

4.7 Original (left) and modified (right) impeller ........................................ 51

4.8 Original (left) and modified (right) impeller blade trailing edge. ............... 52

4.9 Original (left) and modified (right) wear-plate. ..................................... 52

4.10 Computational fluid domain, where D is the inlet diameter. ....................... 53

4.11 Computational grid of the impeller zone. ............................................ 54

4.12 Computational grid applied to the fluid domain. .................................... 55
6.6 Cross-correlation of the hydrodynamic forces and time delay shown as a phase for
6.5 Non-dimensional hydrodynamic forces with flow rate at \( s_g \).
6.4 Non-dimensional hydrodynamic forces for different rotational speed at \( s_g \).
5.11 Variation of pump performance metrics with volumetric flow rate for different im-
5.6 Instantaneous static pressure distribution at \( y = 0 \) (mid-section plane) at \( Q = 198 \) m
5.5 Velocity vectors with flow rate.
5.3 Detailed view of the pressure distribution around the blade tip at different flow rates.
5.4 Velocity vector field at \( y = 0 \) (mid-section plane) for: a) \( \varphi = 0^\circ \), b) \( \varphi = 0^\circ \) at \( Q = 198 \) m
5.2 Instantaneous static pressure distribution at \( y = 0 \) (mid-section plane) for
5.1 Computational model reference frame.
4.15 Output cycle-averaged ratios at \( 198 \) m
4.14 Iterative convergence of scaled residuals.
4.13 Computational mesh in the gap region at section A-A location.
4.12 Variation of pump performance metrics with volumetric flow rate for different im-
4.11 Variation of pump performance metrics with volumetric flow rate for different im-
4.10 Distribution of the vortex core (Q-criterion) in the pump vane for \( \varphi = 270^\circ \) at:
4.9 Pressure coefficient at the inlet face over a complete revolution (\( \bigtriangleup Q = 110 \) m
4.8 Transient pump head over a complete revolution for different boundary conditions [Melzer 
4.7 Static pressure fluctuation at the inlet face over a complete revolution for \( Q = 198 \) m
4.6 Instantaneous static pressure distribution at \( y = 0 \) (mid-section plane) at \( Q = 198 \) m
4.5 Velocity vector field at \( y = 0 \) (mid-section plane) for: a) \( \varphi = 0^\circ \), b) \( \varphi = 90^\circ \), c) \( \varphi = 180^\circ \), d) \( \varphi = 270^\circ \).
4.4 Non-dimensional hydrodynamic forces with gap size at design flow rate and
4.3 Detailed view of the pressure distribution around the blade tip at different flow rates.
4.2 Orbit curves with cycle-to-cycle oscillations from experimental data [Benra, 2006].
4.1 Output cycle-averaged ratios at \( 198 \) m
3.15 Non-dimensional hydrodynamic forces in stationary coordinates at \( s_g = 0.3 \) mm and 
3.14 Iterative convergence of scaled residuals.
3.13 Computational mesh in the gap region at section A-A location.
3.12 Variation of pump performance metrics with volumetric flow rate for different im-
3.11 Variation of pump performance metrics with volumetric flow rate for different im-
3.10 Distribution of the vortex core (Q-criterion) in the pump vane for \( \varphi = 270^\circ \) at: a)
3.9 Pressure coefficient at the inlet face over a complete revolution (\( \bigtriangleup Q = 110 \) m
3.8 Transient pump head over a complete revolution for different boundary conditions [Melzer 
3.7 Static pressure fluctuation at the inlet face over a complete revolution for \( Q = 198 \) m
3.6 Instantaneous static pressure distribution at \( y = 0 \) (mid-section plane) at \( Q = 198 \) m
3.5 Non-dimensional hydrodynamic forces with flow rate and \( \omega = 1475 \) rpm (\( \bigtriangleup s_g = 0.3 \) mm; \( \bigtriangledown s_g = 0.6 \) mm; \( \bigtriangledown s_g = 1.15 \) mm; \( \bigtriangledown s_g = 2 \) mm).
3.4 Non-dimensional hydrodynamic forces with gap size at design flow rate and \( \omega = 1475 \) rpm (\( \bigtriangleup s_g = 0.3 \) mm; \( \bigtriangledown s_g = 0.6 \) mm; \( \bigtriangledown s_g = 1.15 \) mm; \( \bigtriangledown s_g = 2 \) mm).
3.3 Non-dimensional hydrodynamic forces with gap size at design flow rate and \( \omega = 1475 \) rpm (\( \bigtriangleup s_g = 0.3 \) mm; \( \bigtriangledown s_g = 0.6 \) mm; \( \bigtriangledown s_g = 1.15 \) mm; \( \bigtriangledown s_g = 2 \) mm).
3.2 Non-dimensional hydrodynamic forces with flow rate at \( s_g = 0.3 \) mm and \( \omega = 1475 \) rpm (\( \bigcirc \) 17 m\( \cdot \)h\(^{-1} \); \( \bigtriangleup 113.7 \) m\( \cdot \)h\(^{-1} \); \( \bigtriangledown 239 \) m\( \cdot \)h\(^{-1} \)).
3.1 Cross-correlation of the hydrodynamic forces and time delay shown as a phase for 
2.10 Non-dimensional hydrodynamic forces with flow rate at \( s_g = 0.3 \) mm and \( \omega = 1475 \) rpm (\( \bigcirc \) 17 m\( \cdot \)h\(^{-1} \); \( \bigtriangleup 113.7 \) m\( \cdot \)h\(^{-1} \); \( \bigtriangledown 239 \) m\( \cdot \)h\(^{-1} \)).
6.7 Polyharmonic fit of the forces time series with number of revolutions (○ Numerical data; ● Fitted multi-sinusoidal). ................................................................. 84

6.8 Spectra of the non-dimensional forces at the design point: a) $F_x^*$, b) $F_y^*$, c) $F_z^*$. ................................................................. 85

6.9 Spectra of the non-dimensional forces with flow rate: a) - c) $Q = 110\ m^3\cdot\text{h}^{-1}$; d) - f) $Q = 198\ m^3\cdot\text{h}^{-1}$. ................................................................. 86

6.10 Spectra of the non-dimensional forces with gap size: a) - c) 0.3 mm; d) - f) 0.6 mm; g) - i) 1.15 mm; j) - l) 2 mm. ................................................................. 87

7.1 Variation of pump performance metrics with volumetric flow rate for different gaps: a) Pressure Head-Flow rate, b) Average torque about impeller rotation axis, c) Hydraulic efficiency (□ Experimental data; ○ 0.3 mm; ▽ 0.4 mm; △ 0.5 mm; ▶ 0.6 mm; ▪ 1.15 mm). ................................................................. 90

7.2 Variation of pump performance metrics with experimental data [De Souza et al., 2006]. ................................................................. 91

7.3 Variation of the pressure coefficient with the gap size at 198 m$^3\cdot\text{h}^{-1}$ (○ 0.3 mm; ○ 0.4 mm; ▽ 0.5 mm; ▶ 0.6 mm; ▪ 1.15 mm). ................................................................. 92

7.4 Variation of efficiency with gap size for a range of flow rates: a) Variation of efficiency, $\eta$, with gap size, b) Normalised efficiency with normalised gap (□ 110 m$^3\cdot\text{h}^{-1}$; ○ 140 m$^3\cdot\text{h}^{-1}$; □ 170 m$^3\cdot\text{h}^{-1}$; △ 198 m$^3\cdot\text{h}^{-1}$; ▶ 230 m$^3\cdot\text{h}^{-1}$; ▪ 254 m$^3\cdot\text{h}^{-1}$). ................................................................. 93

7.5 Sensitivity of efficiency to gap size: a) Variation of non-dimensional sensitivity factor $K$ with flow rate, b) Variation of efficiency sensitivity factor $\tilde{K}$ with flow rate. ................................................................. 94

7.6 Variation of mechanical power consumption with gap size for various flow rates: a) Mechanical power consumption, b) Mechanical power normalized by a reference power (□ 110 m$^3\cdot\text{h}^{-1}$ ○ 140 m$^3\cdot\text{h}^{-1}$ □ 170 m$^3\cdot\text{h}^{-1}$ △ 198 m$^3\cdot\text{h}^{-1}$ ◊ 230 m$^3\cdot\text{h}^{-1}$ ▪ 254 m$^3\cdot\text{h}^{-1}$). ................................................................. 95

7.7 Sensitivity of mechanical power consumed $P_m$ to change in the gap size $s$: a) Variation of non-dimensional sensitivity factor $\tilde{K}$ with flow rate, b) Variation of sensitivity factor $K$ with flow rate. ................................................................. 96

7.8 Variation of velocity components in the gap region for different gaps at 198 m$^3\cdot\text{h}^{-1}$: a) Axial velocity, b) Radial velocity, c) Tangential velocity, d) Absolute velocity ( △ 0.3 mm; ○ 0.4 mm; △ 0.5 mm; △ 0.6 mm; ▪ 1.15 mm). ................................................................. 98

7.9 Variation of velocity components in the gap region for different gaps and flow rates: a) Axial velocity, b) Radial velocity, c) Tangential velocity, d) Absolute velocity ( △ 0.3 mm, 110 m$^3\cdot\text{h}^{-1}$; ○ 0.3 mm, 254 m$^3\cdot\text{h}^{-1}$; △ 0.5 mm, 110 m$^3\cdot\text{h}^{-1}$; ○ 0.5 mm, 254 m$^3\cdot\text{h}^{-1}$; △ 1.15 mm, 110 m$^3\cdot\text{h}^{-1}$; ○ 1.15 mm, 254 m$^3\cdot\text{h}^{-1}$). ................................................................. 99

8.1 Variation of pump performance metrics at different head variation rates: a) Head with flow time, b) Instantaneous flow rate with flow time, c) Flow rate with flow time averaged over each impeller revolution (□ $\dot{H}=15\ \text{m}\cdot\text{s}^{-1}$; ▽ $\dot{H}=7.4\ \text{m}\cdot\text{s}^{-1}$; △ $\dot{H}=5\ \text{m}\cdot\text{s}^{-1}$; ○ $\dot{H}=3.7\ \text{m}\cdot\text{s}^{-1}$; △ $\dot{H}=3\ \text{m}\cdot\text{s}^{-1}$). ................................................................. 104
8.2 Non-dimensional pump performance with head variation rates: a) Non-dimensional head with non-dimensional flow time, b) Non-dimensional flow rate with non-dimensional flow time (\(\frac{H}{\dot{Q}} = 15 \text{ m s}^{-1} \); \(\frac{H}{\dot{Q}} = 7.4 \text{ m s}^{-1} \); \(\frac{H}{\dot{Q}} = 5 \text{ m s}^{-1} \); \(\frac{H}{\dot{Q}} = 3.7 \text{ m s}^{-1} \); \(\frac{H}{\dot{Q}} = 3 \text{ m s}^{-1} \)).

8.3 Variation of the power with flow time at different head variation rates: a) Hydraulic power, b) Shaft power time (\(\frac{H}{\dot{Q}} = 15 \text{ m s}^{-1} \); \(\frac{H}{\dot{Q}} = 7.4 \text{ m s}^{-1} \); \(\frac{H}{\dot{Q}} = 5 \text{ m s}^{-1} \); \(\frac{H}{\dot{Q}} = 3.7 \text{ m s}^{-1} \); \(\frac{H}{\dot{Q}} = 3 \text{ m s}^{-1} \)).

8.4 Variation of performance metrics at different head variation rates: a) Shaft power, b) Mechanical energy.

8.5 Variation of pump performance metrics with volumetric flow rate for different rotational speeds: a) Pressure Head-Flow rate, b) Average torque about impeller rotation axis, c) Hydraulic efficiency (\(\frac{H}{\dot{Q}} \) Experimental data; \(\omega_1 = 80\% \omega_D \); \(\omega_2 = 90\% \omega_D \); \(\omega_3 = 100\% \omega_D \); \(\omega_4 = 110\% \omega_D \); \(\omega_5 = 120\% \omega_D \); —— system A curve \(h_A\) —— system B curve \(h_B\)).

8.6 Variation of performance metrics at different head variation rates \((h_A, \text{ right}; h_B, \text{ left})\): a) Rotational speed, b) Torque, c) Efficiency (\(h_1 = 35.45 \text{ m s}^{-1} \); \(h_2 = 11.7 \text{ m s}^{-1} \); \(h_3 = 7 \text{ m s}^{-1} \)).

8.7 Variation of performance metrics at different head variation rates: a) Shaft power, b) Mechanical energy (\(h_1 \); \(h_2 \)).

8.8 Variation of performance metrics at \(H = 35.45 \text{ m s}^{-1} \) \((h_A, \text{ right}; h_B, \text{ left})\): a) Head, b) Torque, c) Efficiency \((s_g = 0.3 \text{ mm}; s_g = 0.6 \text{ mm}; s_g = 1.15 \text{ mm}; s_g = 2 \text{ mm})\).

8.9 Non-dimensional coefficients for \(H = 3.5 \text{ m s}^{-1} \): a) Head coefficient \(C_H\), b) Flow coefficient \(C_Q\), c) Power coefficient \(C_W\), d) Hydraulic efficiency \(\eta\) (\(p_1 \rightarrow p_2, \omega_1 = \text{ const}; \omega_1 \rightarrow p_2, p_1 = \text{ const}; \omega_1 \rightarrow p_3, p_1 \rightarrow p_3\)).

B.1 FEM and FVM approaches [Fontes, 2018].

B.2 vertex-centered (left) and cell-centered (right) diagrams [Heinzl, 2007].
List of Tables

2.1 Summary of relevant publications (☐- No, ●- Yes) .............................. 25
4.1 Inlet velocity values ................................................................. 58
4.2 Transient numerical parameters for 78-175 kPa and 1180-1770 rpm ........ 59
8.1 Time step size with rotational speed ........................................... 108
8.2 System curves design parameters .............................................. 108
## Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$b_2$</td>
<td>blade height</td>
<td>m</td>
</tr>
<tr>
<td>$C_p$</td>
<td>pressure coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$C_H$</td>
<td>head coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$C_Q$</td>
<td>flow coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$C_W$</td>
<td>power coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$E_m$</td>
<td>mechanical energy</td>
<td>kJ</td>
</tr>
<tr>
<td>$F_{ref}$</td>
<td>reference force</td>
<td>N</td>
</tr>
<tr>
<td>$F_x$</td>
<td>force along x-axis</td>
<td>N</td>
</tr>
<tr>
<td>$F_y$</td>
<td>force along y-axis</td>
<td>N</td>
</tr>
<tr>
<td>$F_z$</td>
<td>force along z-axis</td>
<td>N</td>
</tr>
<tr>
<td>$H$</td>
<td>pressure head</td>
<td>m</td>
</tr>
<tr>
<td>$\dot{H}$</td>
<td>head rate</td>
<td>m·s⁻¹</td>
</tr>
<tr>
<td>$H_t$</td>
<td>target head</td>
<td>m</td>
</tr>
<tr>
<td>$H_i$</td>
<td>input head</td>
<td>m</td>
</tr>
<tr>
<td>$H^*$</td>
<td>non-dimensional head</td>
<td>m</td>
</tr>
<tr>
<td>$K, \tilde{K}$</td>
<td>sensitivity factor</td>
<td>-</td>
</tr>
<tr>
<td>$K^*$</td>
<td>efficiency sensitivity factor</td>
<td>-</td>
</tr>
<tr>
<td>$\tilde{K}^*$</td>
<td>power sensitivity factor</td>
<td>-</td>
</tr>
<tr>
<td>$M$</td>
<td>harmonic number</td>
<td>-</td>
</tr>
<tr>
<td>$n_{rev}$</td>
<td>number of revolutions</td>
<td>-</td>
</tr>
<tr>
<td>NPSH</td>
<td>net positive suction head</td>
<td>m</td>
</tr>
<tr>
<td>$p(t)$</td>
<td>instantaneous static pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$p_m$</td>
<td>instantaneous mechanical power</td>
<td>kW</td>
</tr>
<tr>
<td>$p_n$</td>
<td>static pressure at n-th revolution</td>
<td>Pa</td>
</tr>
<tr>
<td>$P_{ref}$</td>
<td>reference mechanical power</td>
<td>kW</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>----------</td>
<td>--------------------------------------------------</td>
<td>--------------</td>
</tr>
<tr>
<td>$P_{st,ref}$</td>
<td>static pressure average</td>
<td>Pa</td>
</tr>
<tr>
<td>$P_s$</td>
<td>mechanical power at the rotor shaft</td>
<td>kW</td>
</tr>
<tr>
<td>$P_v$</td>
<td>vapour pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$Q$</td>
<td>flow rate</td>
<td>m$^3$.h$^{-1}$</td>
</tr>
<tr>
<td>$Q_n$</td>
<td>nominal flow rate</td>
<td>m$^3$.h$^{-1}$</td>
</tr>
<tr>
<td>$Q^*$</td>
<td>non-dimensional flow rate</td>
<td>-</td>
</tr>
<tr>
<td>$r_2$</td>
<td>impeller radius</td>
<td>m</td>
</tr>
<tr>
<td>$s_g$</td>
<td>gap size</td>
<td>mm</td>
</tr>
<tr>
<td>$s_{ref}$</td>
<td>reference gap size</td>
<td>mm</td>
</tr>
<tr>
<td>$t$</td>
<td>flow time</td>
<td>s</td>
</tr>
<tr>
<td>$t_n$</td>
<td>single-revolution time</td>
<td>s</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

Growing energy demands, population growth, and new technical innovations are placing significant demands on our planet. These trends are causing species extinction, deforestation and marine habitat destruction, to name a few of the environmental issues caused by human activities, and are leading mankind towards an unsustainable and unpredictable future [Diamond, 2005]. Human activity has increased the concentrations of greenhouse gases (GHGs) to approximately 420 parts per million (ppm) in 2021, up from approximately 280 ppm in pre-industrial times. Although there are a huge variety of GHGs, such as methane (CH$_4$), nitrous oxide (N$_2$O), and other fluorinated gases, the most problematic is carbon dioxide (CO$_2$), as it persists in the atmosphere for thousands of years. The atmospheric concentrations of carbon dioxide, methane and nitrous oxide have increased because of human activities, primarily from fossil fuel emissions and secondarily from net land use [Edenhofer et al., 2011].

According to the World Health Organization, air pollution due to fossil fuel burning is the cause of 7 million premature deaths each year [WHO, 2016]. The international community is attempting to respond to this issue through the United Nations Framework Convention on Climate Change (UNFCC), the Sustainable Development Goals (SDG), and the Paris Agreement to limit global warming below 2°C. SDGs for 2030 [Desa et al., 2016] illustrate how energy production is central to limit global warming below 2°C and could create pathways to meet other SDG targets, like poverty eradication and a reduction in the level of inequality. A recent report from the Intergovernmental Panel on Climate Change (IPCC) [Masson-Delmotte et al., 2018] highlights how renewable energies (RE) and energy efficiency are key to reducing GHG emissions originating from the energy sector which is responsible for approximately two-thirds of total emissions. The International Renewable Energy Agency (IRENA) report of 2019 [Gielen et al., 2019] shows that the adoption of energy efficient and RE technologies, including proper electrification, could reduce CO$_2$ production substantially to meet 90% of CO$_2$ reduction targets. The report cautions that achieving this pathway requires a comprehensive approach that includes a wide range of stakeholders (i.e. the private and public sector, government, research agencies, and communities).

SDGs on clean water and sanitation provide guidelines to address water-related issues. Billions of people still lack basic water services such as safe drinking water and basic washing facilities that are paramount to reduce the spread of pathogens and prevent infections. It is known that
Chapter 1. Introduction

Water and energy are two inextricably linked resources. Within the water sector, wastewater treatment facilities are fundamental to reduce the pollutants of water bodies but require a significant amount of energy to operate continuously. The adoption of energy efficiency policies and RE in water and wastewater utilities (WWU) is gaining momentum to reduce the carbon footprint of treatment plants. Methodologies to reduce the operational costs include load shifting at a time when the energy demand is lower (off-peak periods), adoptions of flow control strategies through variable speed drives, and proper maintenance of the pump components.

Energy represents the second major cost behind labour in wastewater treatment facilities [Lorenzo-Toja et al., 2015, Pabi et al., 2013] and several authors [Nowak et al., 2015] have discussed the possibility of implementing energy balance strategies to achieve energy neutrality or even be suppliers. This could be achieved by taking advantage of the chemical and thermal content of the wastewater bodies through co-generation, biofuel combustion and heat recovery [Krotscheck and Narodoslawsky, 1996]. In order to achieve energy neutrality, it is crucial to identify the various sources of inefficiency to adopt effective measures with the purpose of reducing the energy consumption.

In wastewater treatment plants, pumping is one of the most intensive processes, behind aeration, in terms of energy cost [Maktabifard et al., 2018]. Wastewater pumps have generally non-clog centrifugal designs, with a single or double blade impeller. The design criterion for wastewater pumps is to allow the passage of solid objects through the vane without blockage. As a result, even well-maintained wastewater pumps operating optimally have significantly lower efficiency with respect to other types of centrifugal pumps.

Figure 1.1: Exploded view of the single-blade pump model: wear plate, impeller, volute casing.
1.1 Scope of this thesis

The XFP PE-2 150E CB 1.1, designed and manufactured by Sulzer, is typical of pumps installed in municipal wastewater treatment plants and similar to pumps from other manufacturers (e.g. Xylem’s Flygt, Grundfos). The detailed view of this assembly is shown in Fig. 4.4. The design point of centrifugal pumps is found at the maximum of the efficiency curve, for a specific combination of pressure head and flow rate at constant rotational speed. For all the other operating points, the performance is degraded. In such circumstances, the pump is said to work at “off-design” conditions. However, there could be other possibilities to cause a pump to run at lower performance, such as degradation of physical conditions (e.g. axial gap $s_g$, wear of bearings), that have a significant effect on the energy consumption.

Experimental tests and numerical modelling are the most common solutions to test pump performance under various flow regimes. In recent decades, computational fluid dynamics (CFD) has gained popularity to assess the performance of rotating turbomachinery. However, there is little research on wastewater pumps compared to multi-blade centrifugal pumps. The majority of the studies available focus on the unbalanced radial forces producing radial deflection of the pump shaft that is connected to the impeller. This increases wear and degradation and reduces the durability of bearings and the casing, increasing the energy consumption as well as the total lifetime cost of the installation. There is a gap in the open literature about the effect on the energy performance of the increased gap between the impeller and the wear-plate, and about the impact on the energy consumption when the operating point is switched to off-design conditions as may happen when the pump is operated at partial load conditions.

1.1 Scope of this thesis

The overall aim of this work is to assess the performance of a wastewater pump at off-design conditions. The specific technical objectives are:

- Implement a high-fidelity computational model of the wastewater pump validating its performance with numerical and experimental data from literature;
- Using this computational model, evaluate the performance of the pump when the impeller-wear plate axial gap is increased;
- Assess the impact of transient off-design operating conditions on pump performance.

1.2 Summary

This preliminary chapter highlights the challenges that mankind is facing to address climate change related issues. Measures to provide a more sustainable future across multiple frameworks include new paradigms in terms of energy management, efficiency and optimization. Within water and wastewater utilities, pumping systems are found to be one of the most expensive processes in
terms of energy use. This thesis presents new achievements in terms of understanding the energy requirements of wastewater pumps operating under off-design conditions and is structured as follows:

- Chapter 2 reviews the current policies and future projections of the global energy scenario. It also highlights the relationships between water and energy, and describes strategies to reduce the costs in water and wastewater utilities and hydraulic systems, such as demand-side management. Moreover, an updated review of published data in the area of wastewater pumps and computational methods is presented.

- Chapter 4 describes the fundamental relationships and control techniques found in pumping systems, as well as the main numerical algorithms that were used to replicate the fluid mechanics within the pump channels. The technical data of the XFP PE-2 150E CB 1.1 is also presented. It is also presented the approach adopted to model the geometry and the mesh that are used to define the computational domain of the wastewater pump.

- Chapter 5 reports the transient solution of the computational model for constant rotational speed and various flow rates. The main findings about flow pattern and energy performance are discussed and compared with experimental data.

- Chapter 6 discusses the behaviour of radial forces due to unsteady pressure fluctuations. The forces are obtained at for a range of operating conditions according to different gap sizes, rotational speed and flow rates. The harmonic amplitude and phase with respect to the radial loading is extracted.

- Chapter 7 discusses the effect of the axial gap on pump performance at off-design conditions for various gap and flow rates at constant rotational speed. Non-dimensional coefficients are obtained which describe the drop in performance to allow comparison with similar pumps. An in-depth study of the velocity components within the gap provides an insight of the leakage mechanism.

- Chapter 8 addresses the effects on the performance of adopting pressure-rotational speed control. The performance metrics are plotted for a combination of static pressure and rotational speed with respect to the design value. The transient characteristics relative to two different system curves are examined for various gap sizes. Again, non-dimensional coefficients are obtained.

- Chapter 9 summarizes the main findings and proposes of possible developments of the current research.
Chapter 2

Literature Review

Figure 2.1a shows that the global primary energy supply in 2018 was mostly provided by fossil fuel sources and amounted to 157,063 TWh. Approximately 26,700 TWh of electricity was generated by fossil fuel sources (64%) and the remaining part from nuclear and renewable energies (36%), as shown in Fig. 2.1b [Capuano, 2018, Roser, 2019, BP, 2019].

![World energy data (2018).](image)

(a) Primary energy supply by fuel.  
(b) Electricity generation by source.

Global power demand grew by 3.7% in 2018, and Fig. 2.1 shows that fossil fuels amount to an 85% share in the primary energy supply. The use of fossil fuels grew at a rate of 2.9%, faster than its 10-year averages since 2010, with China, US and India accounting for more than two thirds of the global increase in energy demand [BP, 2019]. The spread in the energy consumption was reflected across all fuels, particularly in natural gas, accounting for almost 45% of growth in global energy demand, followed by renewable energies (14.5%) and coal (3.0%). Renewable power growth was slightly below its historical average but grew more rapidly than any other form of energy, with China leading the way with 45% of the global growth in renewable power generation. Energy production from water sources (e.g. tidal, hydro) accounts for approximately 16% of the world gross electricity production [IEA, 2017b]. Since emissions grew by 2% in 2018, registering the fastest growth of the last seven years and suggesting an inconsistent trend with respect to the Paris Agreement goals, there are clear difficulties to decarbonize the power sector quickly and satisfy the
growing energy demand at the same time. According to IRENA [IRENA, 2019], current and planned policies and technologies are not on track to limit the increase in average temperature, potentially leading to a temperature rise of 3 °C by 2050. Proposed measures to achieve a cumulative emission reduction and reach sustainability goals include increasing the share of renewable energies, reducing the demand through energy efficiency, increasing the electrification for all sectors (especially transport and heat), using distributed energy resources and increasing the share of energy storage systems (ESSs).

The IRENA REmap case scenario [Gielen et al., 2019] suggests that, to achieve a 90% drop in terms of carbon intensity, the share of renewable energy should be more than double with respect to what is currently planned to be installed over the next decades. The network would also require more flexible power schemes, including the integration with variable renewable energy (VRE) systems. Before 2050, it is required that the electricity share grows from 42% to 68% in industry and building sectors, and from 1% to over 40% in the transport sector. To amend that, the allocated and capacity renewable energy supply would rise from around 514 GW of wind capacity and 385 GW of solar photovoltaic (PV) to over 6 TW and 8.5 TW by 2050, respectively, with similar growth rates for geothermal, bioenergy and hydropower. In the industrial sector, concentrated and flat plate solar thermal collectors and heat pumps would be installed to increase the share of renewable energy up to a minimum of 63% by 2050. The transportation sector would see at least 75% of passengers using electric vehicles as hydrogen is implemented as a fuel. All these implementations will require a fundamental change in the nature and the structure of the overall sectors, technology development, behavioural changes and major policy push.

Current difficulties associated with existing variable electricity sources consist in matching supply and demand to avoid blackouts or other issues in the grid. While fossil fuels are able to respond quickly to an unexpected rise in the electricity demand and offer a practical solution in terms of transportability and storage form, renewable energies (e.g. wind, solar) are inherently intermittent due to their sensitivity to meteorological and atmospheric conditions (i.e. weather, seasons variations). However, demand for more flexible grids has become prominent due to growing interest in “green” energy, involving an increase in the demand for additional system storage capacity. Installed capacity doubled between 2017 and 2018 to 8 GWh, with hydro storage accounting for 96.2% worldwide, followed by lithium-ion batteries [WEC, 2019]. It is expected that storage capacity will reach 250 GWh and costs for installed batteries will fall up to 66% by 2030 [Gardner et al., 2016, Ralon et al., 2017].

Continuously growing renewable power generation and storage technology foster additional services including ancillary services, demand response and efficient management of the grid to allow real-time grid operation and demand planning criteria. In this context, distributed energy resources (DERs), electrification and digitalisation of the power sector are paramount to increase the reliability of energy systems in terms of resilience, security and flexibility. Today’s power distribution technologies include centralized power plants that deliver energy one-way through transmission lines and distribution stations, requiring upgrades when new generation assets are installed to satisfy continuously growing energy demand. Moreover, most of the outages occurring in existing
2.1 Demand-side management

Energy storage systems play an essential role in smart grids and are able to respond quickly to power fluctuations generated by intermittent energy sources such as wind and photovoltaic technologies.

![Figure 2.2: Load levelling from peak-load shifting](Yang and Jackson, 2011).
One of the advantages of energy storage systems is the possibility to be stored in the event of excess in the electrical supply, or when the electricity cost is low, to reuse the stored charge when additional power is needed, or when the electricity cost is higher. This management technique is known as “load shifting”, consisting of reducing the energy demand during peak hours, shifting to off-peak times, e.g. night-times and weekends, as shown in Fig. 2.2. This operational target can be achieved through demand-side response measures, such as planning, implementation and monitoring of utility activities to influence end-use of electricity [Gelazanskas and Gamage, 2014]. Examples of such behaviours could be rescheduling activities, switching to on-site electricity generators and switching off sectors of the plant when wholesale prices are high. There exist many other forms of load-shape objectives (e.g. peak clipping, valley filling, etc.), although load management and strategic conservation happen to be the most popular since 1970 [Gellings, 1985]. Load management has been extensively used by electrical suppliers or a fast real-time adaptation of the energy generated to the demand, and hydro power plants used to offer the quickest response. Strategic conservation is a load shape modification resulting from utility-simulated programs for end-use consumption. The term “strategic” requires the utility planner to consider which changes in the energy consumption would occur naturally or deliberately and evaluate the cost-effectiveness to stimulate those actions. Demand response is a useful technique to motivate customers to respond to price changes or energy availability over time, accomplishing load shaping. The most employed DSM programming methods are either incentive-based or price-based, to allow end-users to be involved in load shaping. Because of the large number of operating modes, public owned water supply is an excellent candidate for the application of DSM schemes [Kernan et al., 2017]. Optimization of pumping schemes has been extensively used to minimise the energy consumption during operations [Jowitt and Germanopoulos, 1992, Chase and Ormsbee, 1993, Zhuan and Xia, 2013]. In the past, efforts were directed towards increasing the supply of water to match the demand, which led to water abundance at relatively low cost, with incredible wastefulness. Nowadays, there are several reasons to implement demand management policies other than only for environmental safeguard and sustainability purposes. The main goal is to reduce the energy cost of water supply without compromising public health and service obligations. According to the New York State Energy Research and Development (NYSERDA), there are three key proposed activities that can improve the management of water and wastewater utilities [NYSERDA, 2010]:

- reducing power demand and energy consumption by improving the equipment efficiency and service delivery, reducing the power cost in kWh per cubic meter of water delivered, or waste water treated. Specific examples includes: regular maintenance to reduce leakages and installation of variable speed drives (VSDs) to manage pumps duty cycles;
• managing peak demand and other power system charges by setting operation schedules and consumption charging during peak system load periods. Water and wastewater utilities can reduce their costs by shifting pumping and treating operations to off-peak periods, possibly using automated control systems;
• managing energy cost volatility and improving electricity supply reliability by investing in alternative power supplies, such on-site energy generation which involves the utilization of biogas from anaerobic sludge digesters.

The first two indications are common practice in waste-water utilities. Shifting the operations to night time can help to prevent a pumping network overload, involving higher water pressures and losses according to the demand curve, and therefore an increased energy use. On the other hand, demands for water and electricity are increasing together with carbon emissions. Consequences induced by this relationship can be resumed in the so-called “water-energy nexus”.

2.2 The water-energy nexus

Water and energy are two inextricably entwined resources, with multiple and varied interconnections. More water is demanded as the population grows and it is well known that a lack of supply leads to major health implications. Besides its function of ensuring the existence of every form of life, without substitutes, water is widely used in all stages of energy production, from extraction and mining for fossil fuel production and refining, for power plant cooling in manufacturing industry, to supply waste-water treatment plants. It follows that water is necessary for energy production, and much energy is spent to move water. Although the use of water varies with the technology used, cooling in thermoelectric energy production plants and irrigation are the most water-intensive processes [Bauer et al., 2014]. Depletion of water with little regards to the finite entity of the sources is posing a major concern over future supplies of water and energy. The interrelationships of the water-energy nexus is a relatively new area of research. The effort in optimizing the network for variable energy sources emerged just in the last decades. Some authors [Wang and Zimmerman, 2015, Nair et al., 2014, Tan and Zhi, 2016, Hamiche et al., 2016] have presented a comprehensive state-of-art review of the nexus feature. Statistics for water use are gathered by OECD, Eurostat and the United Nations but often lack the corresponding energy consumption. The diagram presented in Fig. 2.3 shows the interconnectivity of water and energy through many sectors of the U.S. economy. Energy and water flows are shown in green and blue, while estimates are gathered from available data in the period between 2005 and 2011, respectively. The grey flows accounts for dissipated energy, which is one of the largest in the diagram (27 quads/year), and is relative to the inefficient use of energy. The reduction of the dissipated energy through energy efficiency measures is one of the primary technological challenges faced today. Capture and reuse of water from cooling towers, switching to dry cooling systems and recovery of wasted heat with highly efficient power cycles can reduce the amount of water required and increase the efficiency of electricity generation.
On the other hand, non-thermal renewable energies (e.g. PV, wind turbines) require a very little water consumption. Globally, water abstraction mostly occurs from surface or ground aquifers. However, it is normally more convenient to extract water from groundwater reserves, as they are generally of superior quality compared to surface sources and require less treatment. Globally, water abstraction mostly occurs from surface or ground aquifers. However, it is normally more convenient to extract water from groundwater reserves, as they are generally of superior quality compared to surface sources and require less treatment. Technological options to treat or recycle water, such as desalination or wastewater treatment plants, are energy intensive and increase the electricity consumption, generating additional emissions and further contributing to climate change. Figure 2.4 shows how water and energy are interconnected in typical WWTP operations. The green, the yellow and the dotted lines represent the electrical energy, the water flow and backflow of water fluxes, respectively.

Wastewater treatment plants (WWTPs) have been largely employed to reduce the polluted content of urban or industrial discharged water, into designed receiving bodies [Enger and Smith, 2000]. Studies have shown that between the 2 and 3 percent of the global world energy consumption is spent by electric motors to move, process and distribute industrial and municipal clean-water and wastewater [Abelin, 2006]. Recent estimates suggest an increase of the 40% in electricity demand over the years, as Earth’s population is expected to increase to over 9 billion people by 2050 [WEC, 2013, OECD, 2012]. The obvious consequence is that residential and commercial/public service sectors will need more power, as well as all the industrial suppliers which provide energy to urban areas and disposals collection systems, such as water and wastewater utilities. It is clear that improving the energy efficiency of WWU is a primary goal to avoid adding new power generation capacity, as studies have confirmed that electricity production will decrease by 2050 [IEA, 2017a]. It has been estimated that pumping treated water accounts for 70-80% of the total energy consumption in a surface water supply system [Nault and Papa, 2015]. Energy and maintenance costs to run a pump may account for 50-95% of the overall cost during its life time. The already discussed methodologies of demand-side management and other technical strategies can be applied to optimise pumping systems during operations. Some measures may include reducing leaks, lowering the flow rate or the operating pressure to reduce the overall demand. Sometimes, pumping systems are designed to supply maximum demand at all times and pumps are oversized to accommodate future increases in pumping demand, creating an imbalance between pump capacity and system requirements. Oversized pumps result in higher electricity consumption and more frequent maintenance due to wear and tear because of the excess in the energy flow. However, some measures result to be of greater improvement in terms of energy efficiency, such as replacing throttling valves with variable speed control, which may accomplish up to 60% savings in terms of energy consumption [Victoria, 2009].
2.2. *The water-energy nexus*

Figure 2.3: Hybrid Sankey diagram of the U.S. water-energy nexus in 2011 [Greenberg et al., 2017].
2.3 Energy costs in wastewater treatment plants

Wastewater is, by definition, any water discarded after use by humans for domestic, agricultural, commercial or industrial purposes. It may be in the form of liquid or solid wastes, typically composed of 99% water and 1% suspended, colloidal and dissolved solids [Connor et al., 2017]. Wastewater treatment is an energy-intensive process that focuses on organic and nutrient removal to provide reliable protection to public health and the amphibious environment. The treatment process of the wastewater depends on the detrimental content and the originating source. Generally, this mechanism involves collecting wastewater in a septic tank and transferring it to a treatment plant where the pollutants are removed. In developed countries, water treatment is responsible for the largest electricity consumption in the urban water cycle. According to USA EPA [Fitzsimons et al., 2016], wastewater treatment accounts for 1% of the total energy consumption. In USA and Europe, about 3% of the total electricity share is consumed by water and wastewater utilities, excluding the additional embedded energy to improve and maintain the infrastructures. An overview of the typical wastewater treatment processes is reported in Fig. 2.5. Electricity input is considered one of the most significant costs for water and wastewater utilities and can be up to 30% of the total operating expenditure [Abelin, 2006]. The amount of energy consumed within the various stages depends upon size (population equivalent, organic, or hydraulic load), age and location of the plant, population served, treatment types, water discharge quality, operating costs and experience of workers and managers [Foladori et al., 2015, Bodik and Kubaska, 2012, Plapally, 2012]. In developing countries, such as Bangladesh, where water and wastewater utilities are usually owned and operated by the central government, the electricity cost can be over 40% of the total [Van Der Berg and Danilenko, 2011] and can have significant impact on the country’s budget. In India, for example, water supply is the greatest cost among all municipal services [Mukesh, 2000].
Moreover, it is estimated that by 2030, water and wastewater utilities will experience a 40% discrepancy between water demand and supply due to world population increase and water scarcity due to extreme events (e.g. floods and droughts) [UN, 2007, Liu et al., 2012]. Improving the energy efficiency of water and wastewater utilities is the bottom line to reduce the demand of adding power generation capacity and minimize operational costs, emissions and pollutants. The overall electricity use per unit of water (or wastewater) processed for end-use is expressed as kWh/m$^3$. It is misleading to generalize and compare system-level performance between utilities, as they are significantly affected by operating conditions (e.g. daily flow distribution, elevation, etc.), and treatment technology. For example, advanced wastewater treatment with nitrification could be up to 3 times as energy intensive as trickling filter treatments due to additional pumping requirements. A survey of publicly owned treatment plants in the USA yields an aggregated value of the energy consumption for typical facilities: 0.252 kWh/m$^3$ for trickling filter systems, 0.349 kWh/m$^3$ for activated sludge, 0.407 kWh/m$^3$ for advanced wastewater treatment without nitrification and 0.505 kWh/m$^3$ for advanced wastewater treatment with nitrification [Goldstein and Smith, 2002]. It is also shown that unit electricity consumption for secondary treatment decreases with increasing plant size, as shown in Fig 2.6a. Different methodologies for energy use accounting have been adopted to calculate the costs and the environmental burden of water utilities and wastewater treatment plants. For example, in most medium and large plants using conventional activated sludge (CAS) treatment, aeration is the highest energy consumer component. According to Mamais at al. [Mamais et al., 2015], in medium and large wastewater treatment plants with CAS systems, the total electric consumption is split on aeration (60%), sludge treatment (15-25%) and secondary sedimentation and recirculating pumps (12%), as displayed in Fig. 2.6b.
Chapter 2. Literature Review

(a) Unit electricity cost with million gallons per day each treatment process [Goldstein and Smith, 2002].

(b) Energy share in a typical wastewater treatment plant [Walther, 2009].

Figure 2.6: Wastewater pumping is typically 12% of the energy consumption or approximately 0.05 kWh/m³.

In municipal wastewater plants, secondary and advanced treatment stages are more energy intensive than raw wastewater collection and pumping [Pitas et al., 2010, Kneppers et al., 2009]. Wakeel et al. [Wakeel et al., 2016] carried out a review addressing the status, research gaps and possible improvement measures in the water sector. It is shown that water supply and wastewater services are energy intensive globally owing to old infrastructures and technologies. The review reported that three approaches are generally used to calculate the energy consumption in the water and wastewater utilities: the bottom-up (life-cycle analysis or LCA), the top-down (input-output analysis or IOA) and hybrid approaches. Standard life-cycle analysis is a standardized method with a “cradle-to-grave” analysis through all phases of life of a process, or service, and has been used by several authors to estimate and reduce the energy cost and appraise the environmental burden such as global warming issues [Stokes and Horvath, 2006, Nair et al., 2014]. The great advantages of LCA is the possibility to appraise the various indirect responses as a single category, as new global policies impose mandatory environmental assessments. Nevertheless, this technique has shown some drawbacks, such as being data-intensive and leading to gaps and omissions inevitable to some degree [Landu and Brent, 2006]. Input-output approaches are used for assessing energy, environmental emissions, and water and ecological footprints. However, price differences and low level of detail on sector resolutions are major weaknesses of this top-down approach. Hybrid approaches tend to be more accurate than standard LCA and IOA but can be more data-intensive. Racoviceanu et al. [Racoviceanu et al., 2007] used life-cycle inventories to identify wastewater treatment processes with the greatest environmental impact so that they can be targeted for improvement. The authors found that in water supply facilities operations are the dominant contributor to overall energy use and greenhouse gas emissions, targeting pumping as the major energy (64%)- and GHG (90%)- intensive process. It is suggested that increased pumping efficiency, reduced leakage in the distribution system, electricity monitoring and minimisation
2.3. Energy costs in wastewater treatment plants

of the chemicals can significantly enhance system performance and meet human health requirements. Molinos-Senante et al. [Molinos-Senante et al., 2014] carried out a relatively new benchmarking approach known as radial Data Envelopment Analysis (radial DEA), which is a parametric method able to consider a multiplicity of inputs, outputs and units which enables the isolation of specific factors in order to save costs. Results are very sensitive to the system inputs and outputs. LCA and DEA analysis can also be coupled both to assess operational targets and environmental implications at the same time. Lorenzo-Toja et al. [Lorenzo-Toja et al., 2015] published an extensive study performed on Spanish wastewater treatment plants using LCA and life cycle costing (LCC). The latter is another cradle-to-grave approach, as LCA, but it looks at the direct monetary cost involved with a product or service and not environmental impact. They found that environmental impact of treatment facilities does not always depend on the plant size and efficiency but operating behaviour instead, so that even smaller and less efficient plants can show lower emissions if managed properly. Moreover, they found that labour cost is the highest budget item for water and wastewater utilities, followed by energy consumption [Lorenzo-Toja et al., 2015, Pabi et al., 2013].

There are many ways to improve water and wastewater utilities performance. For example, Jonasson [Jonasson and Ulf, 2007] shows that continuous benchmarking studies in Austria led to reduced energy costs of about 30% since 1999. The study analysed wastewater treatment plants of similar size in Sweden and Austria and found that Swedish installations uses roughly 45% more electrical energy than Austria. One of the reasons for the lower consumption is that Austria has carried out benchmarking studies for many years. Repeated energy benchmarks encourages competitions between wastewater treatment plants operators and decreased consumptions are achieved by the need for improvements. Moreover, the author found that similar comparisons are difficult for all countries, especially for those recently developed. This is due to incomplete data capture. For instance, Chinese wastewater treatment plants, where often sludge is transferred to a third party without further information on the energy consumption. Gu et al. [Gu et al., 2017] stated that this exclusion can explain the reason for the apparent low energy consumption for wastewater treatment plants in China. The same authors have carried out a thorough review of the wastewater treatment plants energy requirements. They focused their research on understanding the energy consumption through a wide literature review, pointing out that efficiency optimization and theoretical analysis are the most recent advancements. Funamizu et al. [Funamizu et al., 2001] published a study on the reuse of heat energy in Japanese wastewater plants, implementing thermal recovery through heat pumps. The work suggests that water thermal content can be used for secondary purposes, such as heating nearby facilities. Some authors [Meyer, 2000, Kempton, 1988, Hopp and Darby, 1980] already identified the advantages of installing hot-water conservation devices. Meyer et al. [Meyer, 2000] analysed the feasibility of such appliances in Johannesburg peripheries, where the population of low-density houses used four times more water than people living in high-density housing. Hopp et al. [Hopp and Darby, 1980] stated that residential heating needs for a family of four in USA can save 46-62% of water heating by using thermal energy recovery from neighbourhood sources (e.g. cogeneration plants). More recently in Europe, heat recovery systems are operating in Switzerland, Scandinavia and Austria. The estimated amount
of thermal energy that could be recovered with heat pumps from Austrian wastewater treatment plants is 450 GWh/y [Venkatesh and Brattebo, 2011, Pandis Iveroth et al., 2013, Nowak et al., 2015]. Similar solutions have been recently proposed by Awe et al. [Awe et al., 2016] for Ringsend’s wastewater treatment plants in Dublin, involving the reuse of this content in neighbouring facilities, or to supply the plant itself. According to this view, one of the targets of wastewater plants should be the reduction of the net energy input. Although wastewater treatment plants are actually energy consumers, they can become suppliers in the future. The organic fraction usable as energy source in wastewater is measured by the chemical oxygen demand (COD), which indicates the amount of oxygen ($O_2$) needed to oxidize the organic material in a solution. Some technologies allow the extraction of methane ($CH_4$) from both biodegradable and refractory fractions of sludge, at the cost of additional energy input [Speece, 2008]. Nowak et al. [Nowak, 2003, Nowak et al., 2011] studied the energy gain by using methane (CH4), which has been degraded biologically through anaerobic treatments. In a subsequent work [Nowak et al., 2015], the same author shows that by adding “biowaste” to the sludge digester, the electricity production can grow on-site up to 180% of its own energy demand, and the thermal recovery by cooling water through regeneration can save up to 45% of the electricity input. However, the overall energy demand for a wastewater treatment plant must be studied case by case, as the literature review indicates that it is difficult to establish the overall costs \textit{a priori} using mathematical models.

Research in the last decades has been focused on energy self-sufficient wastewater treatment plants, which can self-generate their own energy requirements and even be suppliers [Svardal and Kroiss, 2011]. Environmental concerns driven by climate change, fossil fuel consumption and increasing energy costs are shifting the efforts towards a more sustainable use of water and wastewater energy potential. It is estimated that while wastewater treatment requires up to 0.6 kWh/m$^3$, the energy content of such bodies of water is up to 10 times that required for treatment [Gude, 2015, McCarty et al., 2011]. Wastewater treatment facilities could reduce the energy input more than 30% by applying energy efficiency measures, using energy generated from renewable sources or through thermal recovery [Edward, 2004]. Englehardt et al. [Englehardt et al., 2016] presented the net-zero water concept to study the feasibility of water reuse applications to address water shortages and energy demand from water and wastewater utilities through the use of the emerging direct potable reuse (DPR) technology. However, current implementations of more sustainable policies are limited to research studies, as regulatory standards and technologies that are not yet commercially available to allow for wider adoption. Bertanza et al. [Bertanza et al., 2018] found that plants equipped with sedimentation and anaerobic digesters are more likely to achieve self-sufficiency, but upgrading existing plants could be more challenging than building new ones [Gao et al., 2014]. Moreover, there is little public acceptance of potable reuse of water after purification, especially from older generations, despite increased perception of the value of water in the general public. Maktabifard et al. [Maktabifard et al., 2018] carried out an extensive review to indicate the actions to be taken on achieving energy neutrality in wastewater treatment plants. A major step include reducing the energy consumption in AD by switching off blowers 25% of the time using a control system, as aeration represents the largest share of electricity consumption. According
to the literature, between 25% to 56% of the operating cost in wastewater treatment plants are associated with energy, and more than half is due to aeration [Castellet-Viciano et al., 2018]. Furthermore, the chemical content in raw wastewater could be used for on-site energy generation by coupling AD and biomethane production with combined heat and power engines. Additionally, bioelectrochemical systems (e.g. microbial fuel cells) to capture the energy potential directly from dissolved organics into electricity are considered as a promising technology. With modern conversion techniques, only about 30-40% of the CH$_4$ potential is converted into electricity, the remaining is given off as heat [EPA, 2007]. In this context, energy efficiency and improved managing programs plays a fundamental role to achieve energy and water neutrality in water and wastewater utilities. However, a detailed explanation on modern techniques addressing potential improvement in the process itself is redundant for the purpose of this thesis. One of the aims of this research is the definition of the energy requirement for specific devices such as wastewater pumps, which are the main vehicle for the water transmission outside and inside wastewater utilities.

### 2.4 Hydraulic pumps energy costs and saving opportunities

Pumps are fundamental devices commonly used to move fluids across a wide spectrum of applications. Of those belonging to the sub-class of rotating turbomachinery, centrifugal pumps are commonly employed in water distribution systems and wastewater utilities, as well as many other civil, industrial and farming applications. Their role is to convert the angular momentum from a “prime mover”, typically an electric motor, to hydraulic energy that allows a fluid to move through a pipe by overcoming the energy losses and the topological challenges within a pipeline system. Between 2% and 3% of the total world electricity consumption is consumed by electric motors to move, process and distribute industrial and municipal water and wastewater [Abelin, 2006].

In water distribution systems, pumps are the highest energy consumer as they account for more than 85% of the electricity share [Burton, 2017]. However, pumps cannot be considered as a standalone equipment but as a part of a system in which they operate. There exist many factors to be considered, such as industry characteristics, demand profile and electricity tariffs. In clean water extraction and distribution, there are several aspects that identify uneven energy requirements, depending on the source. Groundwater provides up to 40% of the global drinking water, and in many countries can be the most significant fraction of the water supply system, (e.g., Denmark 99%, Mexico 95%) [Scott, 2013, Shah, 2009]. However, there is a substantial difference in the energy consumption in different regions. An example is found in the USA: in California, up to 7% of the energy is spent in pumping ground water, while the national average is about 1-2% [Bennet et al., 2010, Schwarzenegger, 2006]. Reasons for this discrepancy are related to the topology of the local area, volume, and depth of the source body and technology employed. In surface water industry, the energy consumption is a function of the pipeline structure (e.g. length, diameter, friction), therefore, it depends on the actual distance between suppliers and utilities [Reardon and Newell, 2010]. Studies have reported that groundwater extraction can be up to 30% more energy intensive than surface water on a unit basis [Goldstein and Smith, 2002].
In wastewater treatment plants, pumping may account for 25-50% of the total energy input, being often the higher consumer after aeration [Maktabifard et al., 2018, Europump, 2004]. Sewage centrifugal pumps of the single or double blade type are widely employed in wastewater treatment plants due to the large flow passage (usually called “ball passage”), or generally for all purposes where a large portion of the fluid is composed by fibers or solids, to avoid clogging. In general, pumps are oversized for their applications as their choice is based on future demand forecasts, and consequently often operates with throttled discharge valves. As a result, pumping operational conditions are mostly inefficient, causing increased energy consumption. Although many types of system arrangements and pump combinations have been explored to satisfy water distribution systems’ needs, none of them can operate efficiently using throttling valves. Energy audits reveal that the price to purchase a pump is about 1-5% of the life-cycle cost (operation, maintenance) to run it [Jenkins and Wanner, 2014]. Moreover, pumps performance deteriorates over time, causing additional energy consumption. According to WERF [Crawford, 2009], the overall efficiency of the pumping system, called “wire-to-water’ efficiency, is calculated by the product of single component efficiencies: the electric motor, the pump, and the drive and control systems efficiencies. Other reasons for inefficiency are caused by fluid dynamics unsteadiness such as turbulence and recirculating flow within the pump vanes, which highlights the challenge of matching the best efficiency point (BEP). Furthermore, pumping operations are highly influenced by topological factors, treatment level, distance that the water travels and end use. For instance, pumping 1 m$^3$ of water in Melbourne (Australia) requires 0.09 kWh, compared to 2.3 kWh in southern California [Olsson, 2012]. Energy use for pumping can be reduced by means of improving the efficiency of such systems, including primary pumping, recirculation and sludge pumping.

It is estimated that around 20% to 50% of the power consumed can be saved by varying the speed of the pumping system, which eliminates the need for throttling valves [Gaudani, 2009], and by selecting components and dimensioning the system appropriately [Tamminen et al., 2013, Lu, 2016].

**Figure 2.7:** Pumping system with pump drive train [Arun Shankar et al., 2016].
A solution to overcome the problem is to couple constant speed pumps with variable frequency drivers (VFDs), so they become variable-speed pumps (VSPs). The typical pumping system using a variable frequency drives is illustrated in Fig 2.7. The AC supply feeds the transformer and the VFD changes the frequency of the electric motor that supplies the pump. In industries this system is conventionally used to let an operator change the pump’s output (flow) rather than as part of an automatic closed loop control system. It is well known from literature [Desa et al., 2014, Maheswaran et al., 2012, Dave et al., 2012, Europump, 2004, Neuberger and Weston, 2012, Pemberton, 2005] that VSPs provide improvements for hydraulic pipeline performances. The minimization of the energy costs in WDSs is not a new argument. Detailed discussions were already made in early 90s by ACEE [ACEE, 1991], and by Ula [Ula, 1991] in his M.Sc. thesis, providing also an early detailed description of the technique. Their main conclusions state that VSPs are devices which can effectively reduce the energy input, improving wire-to-water efficiency. However, the coupling with VFDs is not always attractive, as the energy savings apply only to a limited flow regime. However, some of the energy conservation strategies that can be achieved through variable speed control pumps are here presented [Viholainen, 2014, Olszewski, 2016]. DeBenedictis et al. [DeBenedictis et al., 2013] developed an approach based on a pump coupled with a variable frequency drive and a programmable logic converter (PLC) to improve the overall system efficiency. Optimal control of pumping systems is used as a more efficient operational methodology, conventionally for pumps operating in parallel by choosing the number of those to be staged/de-staged in a typical duty cycle [Ma and Wang, 2008]. The optimization algorithms generally set the energy cost (power required) as the objective function to be minimised, with a desired output (i.e. flow, head, volume of the tank) as constraints [Sycha, 2004, Ibarra and Arnal, 2014]. Zhang et al. [Zhang et al., 2012] used a neural network algorithm for a pump system scheduling module to reduce the energy consumption in a wastewater treatment plant. The pump system performance uses a data-driven model to determine pump configurations and pump speed control settings instead of performance curves and affinity laws. Some changes in the optimal control implementation for VFD involve the coupling with the pipeline. Some of the actual models examined [Da Costa Bortoni et al., 2008, Yang and H., 2010] do not take into account changes in the system curves, or the fact that system curves are different for each pumping system. Moreover, they do not take into account that there are changes in the pump efficiency based on the speed reduction [Marchi et al., 2012].

Figure 2.8 shows the advantages of adopting variable frequency drives. In Fig. 2.8, the speed variation for a given nominal static head allows to move the point A, at speed \( N_1 \), to a different operating point \( A' \), with speed \( N_2 \), by keeping the same efficiency at nominal speed. As power and torque are proportional to the cube and the square of the speed, respectively, halving the velocity results in a reduction to 12.5% of the motor power consumption. The advantages lies in altering the electric frequency of the supply motor, by keeping constant the voltage to frequency ratio, in order to move along the pump’s characteristic curve as the speed is varied, resulting in a transmitted torque at higher efficiency. Moreover, electric motors are never 100% efficient: the maximum efficiency is usually between 70% and 96% of the rated load depending on the motor size, and can be lower when operating at part load [Kaya et al., 2008].
Some authors [Sarbu and Borza, 1998, Marchi et al., 2012, Sarbu and Borza, 2003] show that the prediction of the VSP total efficiency at high loads can fail, and it can drop more than expected even for small speed rate reductions, for those systems which are characterised by large static head. An example is provided in Fig. 2.8, where for a large static head there is a loss in efficiency by moving the operating point from B to B', resulting in a lower efficiency compared with the nominal speed in B. In addition, degradation of the physical conditions of a pump during its lifetime may have a significant impact on the energy usage. Several authors assessed the actual operational global energy consumption in wastewater treatment plants and identified inefficiencies in pumps and aeration systems [Torregrossa et al., 2016, Krampe, 2013]. The overall pump condition should be monitored indirectly by noting the average flow rate and energy consumption data over a period [Torregrossa et al., 2016] and regular maintenance undertaken (typically once or twice per year) to reduce leakage in mechanical seals [NYSERDA, 2010]. The emphasis on seals in maintenance protocols reflects the fact that the vibration induced by the impeller radial thrust can be severe due to the asymmetric nature of the impeller in wastewater pumps. This radial loading can significantly shorten the life of bearings and seals thus affecting the device performance [Okamura, 1980, Aoki, 1984].

The review on hydraulic pumps’ energy costs highlights many opportunities to improve the system efficiency for cost and energy savings, that can be summarised as follows:

- Replacing throttling valves with variable speed controls driven by efficient electric motors with appropriate component selection;
- Optimize pump system design and size (pump, motor, VFD) to allow operation near the best efficiency point;
- Optimizing the piping system by reducing bends, valves, bypass flow loops and fittings and increasing pipe diameter to reduce friction;
- Replace damaged pump components, cracks and leakages by performing active and frequent maintenance.

2.5 Computational methods for centrifugal pumps

Experimental model testing is the most common solution adopted by manufacturers to measure pump performances under various flow regimes. The usual setup consists of the pump, the electric motor, a pipeline with flow control valve, shaft torque meter, pressure transducer and flow meters at the inlet and at the outlet to measure the pressure difference and flow rate through the pipes. Particle Image Velocimetry (PIV) is used to map the instantaneous velocity distribution of flow particles within the pump vanes. This technique often implies a stationary viewpoint (reference frame) to observe the rotating flow, hiding the relative flow structures (e.g. turbulent structures) within the primary flow [Adkins and Brennen, 1988]. Some authors [Sinha and Katz, 2000, Feng et al., 2009b, Zwingenberg and Benra, 2006] applied PIV to investigate the unsteady flow structure in radial pumps. The potentialities of such experimental technique are improved when coupled with Laser Doppler Velocimetry (LDV) [Akhras et al., 2004, Feng et al., 2009a]. In order to capture the unsteady flow from a rotating reference, [Stickland et al., 2002] used a de-rotator to enable the PIV image system to move at the same speed of the rotating impeller. As one can imagine, setting up an experimental rig to evaluate pumps performance for research purposes is often costly and highly time consuming.

On the other hand, a theoretical approach is preferred to represent the fluid dynamics within the pump channels. Recent reviews on the state-of-art have shown a widespread tendency to take advantage of growing computer power to predict pump performance [Keck and Sick, 2008]. Numerical analysis is a useful method to assess the behaviour of a geometry model and compare the prediction with experimental data. Technology advancements led to the implementation of Reynolds Averaged Navier-Stokes (RANS) equations into numerical simulations, initially with finite volume methods, lately extended to unsteady simulations until today, when Unsteady-RANS (URANS) are solved with advanced turbulence models as k-ε and k-ω, Shear Stress Transport (SST) and Large Eddy Simulations (LES). Codes make typically use of three methods to simulate the flow behaviour within the pump vanes: the Multiple Reference Frame (MRF, or Frozen Rotor), the Mixing Plane approach, and the Sliding Mesh method. They mainly differ in the coupling between interfaces that separate rotor and stator. In the MRF and mixing plane methods, the position of the rotor is fixed, and the mesh does not move. For this reason, it ignores the relative motion of the fluid zones with respect to each other, using a circumferential averaging technique to interpolate the flow variables at the interface. The sliding mesh, as the name suggests, allows the different mesh zones to move relatively to each other at each time step of the numerical simulations. For this reason, it is the most accurate computational method for simulating rotating flows, although it is also demanding in terms of computational resources [Gulich, 2008]. For the reason of being more accurate, it was chosen to be used in the simulations reported in this work.
With CFD, variations of the design parameters can be incorporated in the numerical model to assess the fluid dynamics within the pump channels and carry out the performance metrics, without the necessity to manufacture the actual pump and setup an experimental rig. For instance, some authors [Bacharoudis et al., 2008, Agnastopoulos, 2006] have accomplished numerical simulations to address variations in the outputs parameters (e.g. head, efficiency) by changing the impeller design configuration [Zhou et al., 2013]. [De Souza et al., 2008] combined Design Of Experiment (DOE) method with CFD, to optimize the impeller of a single-blade wastewater pump, developing an advanced design with improved solid handling ability and higher efficiency. In a subsequent work, [De Souza et al., 2010] applied the constant-velocity methodology to design a volute for a single-blade pump, using computational methods to find good agreement between the theoretical velocity field and the numerical calculations. The use of CFD has been widely accepted as a strategy to estimate the flow behaviour in pumps in a wide range of operating conditions, such as: off-design, cavitation, interaction effects with different components, performance in turbine mode etc. [Shah et al., 2013]. Although there are plenty of numerical works for multi-vane centrifugal pumps for clean water purposes, the same is not true for single-blade pumps. These pumps have unconventionally shaped impellers and asymmetry of the vane, which are required to allow the passage of large objects and impurities. The flow development is highly sheared and presents strong three-dimensional effects, emphasizing the need for CFD to analyse the fluid dynamics within the rotating region.

Many of the research findings in the open literature is focused on the fluid-structure interactions due to unsteady hydrodynamic forces and flow induced vibrations. Of the forerunners who investigated this peculiar kind of pumps there are Agostinelli et al. [Agostinelli et al., 1960] and Okamura [Okamura, 1980], whose study focuses on the oscillations induced by the impeller radial thrust, which shortens the life of bearings and seals, affecting pump efficiency. Although those are fully experimental studies, they investigate behaviour of typical single-vane pumps, highlighting the differences with multi-vane casings, and providing the first case study for following assessments. Aoki [Aoki, 1984] carried a similar investigation using sensors to measure the instantaneous oscillations induced by inter-blade pressure difference. These studies provided an insight to the distorted pressure distribution field caused by the tongue-volute interaction at a time when computational methods were not so common due to the limitations of computing power. Another experimental approach on a single-blade pump is provided by the study of [Melzer et al., 2020], who measured the flow rate fluctuations amplitude and phase using an electromagnetic flow meter. [Benra et al., 2005] compared velocity measurements obtained with PIV with CFD using SST k-ω turbulence model. Some authors [Benra, 2006, Benra and Dohmen, 2007, Savillius and Benra, 2006, Nishi and Fukutomi, 2014a] investigated fluid-structure interactions, comparing experimental data with CFD, where k-ε and SST k-ω turbulence models were addressed to solve the numerics in the rotating regions. Despite the similarities of those studies, each one clearly shows the progresses achieved by software developers coupling the finite volume and finite elements solvers for unsteady analysis. In particular, Nishi and Fukotomi [Nishi and Fukotomi, 2014a], examined the influence of the blade tip outlet angle on the radial thrust, showing that the fluctuating component
of the radial force increases for higher blade angles at high flow rates. Majidi [Majidi, 2005] investigated blades-tongue interactions in a multi-blade centrifugal pump, finding that larger amplitude of pressure fluctuations and stronger vibration occur in the channels as the blade approaches the tongue. [Parrondo-Gayo et al., 2002] investigated the tongue-gap interactions at off-design conditions, finding that the pressure fluctuations at any point the volute result from the superimposition of the perturbation induced by the passage of the blade in front of that point and in front of the tongue. More recently, Pei et al. [Pei et al., 2012a, Pei et al., 2012b, Pei et al., 2012c, Pei et al., 2012d, Pei et al., 2013a, Pei et al., 2013b, Pei et al., 2014c, Pei et al., 2014a, Pei et al., 2014b] led a thorough study to assess the different phenomena occurring within a single-blade pump, through the combination of experimental tests conducted in their research laboratories and numerical simulations computed by coupling fluid-dynamics and fluid-structure solvers. This research consisted of a wide overview concerning vibrations induced by pressure fluctuations, especially at off-design conditions, where those phenomena have been found to be more relevant. One way and two-way coupling methods were used to estimate the hydrodynamic force for different meshes and various flow conditions. These approaches are based on a partitioned method whether the flow field is affected by structural deformation or vice versa (one-way) or both are solved at the same time step [Ahamed et al., 2017]. The simulations show a bigger amplitude variation of the hydraulic force when coupling effects of the fluid-structure interaction are considered and from the normal and tangential stresses on the blade surface caused by the unsteady pressure field around the single-blade impeller. During off-design operations, the absolute maximum of these forces is lower for lower flow rates but higher on average for higher volume flows [Savillus and Benra, 2006]. The higher pressure fluctuations are observed around of the blade outlet when the tip crosses the volute tongue, while the turbulent intensity is stronger on the suction side. These effects are intensified with leakage flow, which is typical at part-load in single-blade impellers [Pei et al., 2014a]. A similar investigation was carried out by Nishi and Fukotomi [Nishi and Fukotomi, 2014a, Nishi and Fukutomi, 2014b, Nishi and Fukutomi, 2015], who studied how the blade outlet angle affects radial and axial thrust and leakage. They found out that the total thrust increases with the flow rates and both axial and radial thrusts are larger for higher blade outlet angles. Similarly, leakage flow rate increases at higher flow rates and larger outlet angles because of the increased static pressure around the impeller and higher pressure difference between the shroud rear and front. Similar results have been found in the current study, as discussed in Chapter 7.

The literature shows a scarce presence of pure numerical works concerning single-blade pumps, and a large number of the already reported publications were conducted by solving two-dimensional grids. Kurilla et al. [Kurilla et al., 2019] compared three different single-blade impellers with laboratory measurements using two different turbulence models with two frame change options (frozen rotor, stage). They found that an unsteady method was more reliable to predict pump performance, although some discrepancies were found as the numerical models do not take into account mechanical friction losses such as in bearings, resulting in an under-prediction of the simulated head, and an over-prediction of the efficiency. Melzer et al. [Melzer et al., 2018] carried out a numerical study to investigate the cycle-to-cycle pressure fluctuations and compared the results with
experimental data. They concluded that overestimation in the performance metrics, specifically in the head, can be assumed to be due to the statistical nature of turbulence models, suggesting that scale adapting simulations (SAS) combined with SST could be beneficial for prediction of the transient behaviour of single-blade pumps. Keays and Meskell [Keays and Meskell, 2006] were one of the first to implement a numerical methodology by using commercial CFD solvers (e.g. Fluent and CFX) to report single-blade pump performances, recreating the characteristic curves of head-discharge and power consumption by using k-ε and Reynolds Stress Model (RSM) turbulence models, finding good agreement with experimental data. Secondary effects such the influence of the axial and radial clearance between the blade tip and the volute walls are known to be detrimental for the energy performance of single-blade and multi-blade centrifugal pumps [Wood et al., 1965, Zhou et al., 2003, Lei et al., 2015]. It is worth mentioning that radial and axial gaps between the impeller and the casing induce different phenomena, with different experimental and computational challenges. [Zemanová and Rudolf, 2020] carried out a literature review of the current state of art regarding flow inside sidewall gaps of hydraulic pumps and turbines. [Barrio et al., 2008] studied the volute-tongue interactions in a multi-blade centrifugal pump for different impeller-tongue radial gaps. [Stickland et al., 2000] and [González et al., 2006] found that reducing the radial gap from 15.8% to 10% of the impeller radius increases the maximum pressure amplitude of about 50%. The authors also claim that some of their findings could be generated using similarity laws, up to 0.7 Qn. [Blanco et al., 2005] investigated a similar case, finding good agreement between numerical and experimental data. In this study, the authors also found that as the flow rate increases, the stagnation point around the tongue shifts from the diffuser side to the volute side. [Will et al., 2012] carried out a numerical study of the axial gap in a seven-blades centrifugal pump using k-ω SST model, analysing the pressure distribution of the junction ring in the inlet region. [Su et al., 2016] studied leakage in a multi-blade pump as turbine (PAT), finding that leakage flow reduces the volumetric efficiency, reducing the performance as the flow rate increases. [Wu et al., 2017] investigated the velocity profiles across the gap, finding that the negative value of the radial velocity results in an inward radial flow, causing leakage into the inlet through the impeller wear-ring. [Engin and Gur, 2001] conducted an experimental study on the axial tip clearance ratio for centrifugal pumps working with fine slurries, conditions which are very similar to to what occurs for wastewater pumps. In their validated work, the measured head and efficiency decrease up to 20% causing a drop in the pump general performances. Liu et al. [Liu and Tan, 2018] investigated axial gap leakage in a PAT in pump mode, finding that the presence of the axial clearance causes unstable flow structures and complex vortices in the passage, decreasing head and efficiency. More in-depth studies focusing on the hydrodynamics of tip leakage vortex [Liu and Tan, 2019a, Liu and Tan, 2019b] found that the extent of flow instability is determined by the oscillation of tip leakage vortex and blade number. Hao et al. [Hao and Tan, 2018] found that tip leakage, especially for un-symmetrical clearances, affects pump cavitation performance, magnitude and direction of radial force. De Souza et al. [De Souza et al., 2006] investigated the change in performance due to axial clearance at 1% and 2% blade heights, highlighting the severe computational effort due to the
scale of the flow being resolved and the significance of the unsteady flow phenomena within the localised region. Although it does not clearly quantify the performance drop from an energetic point of view, it is one of the few and earliest studies addressing the effects of axial tip-gap clearance in single-blade pumps. An overview of the literature is summarised in Table 2.1.

Table 2.1: Summary of relevant publications (○- No, ● - Yes).

<table>
<thead>
<tr>
<th>Scope and Methodology</th>
<th>Experimental</th>
<th>Numerical</th>
<th>Authors</th>
</tr>
</thead>
<tbody>
<tr>
<td>Review</td>
<td>●</td>
<td>●</td>
<td>[Keck and Sick, 2008]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>○</td>
<td>[Shah et al., 2013]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>○</td>
<td>[Ahamed et al., 2017]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>○</td>
<td>[Zemanová and Rudolf, 2020]</td>
</tr>
<tr>
<td>Experimental tests - PIV</td>
<td>●</td>
<td>●</td>
<td>[Feng et al., 2009a]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Feng et al., 2009b]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Stickland et al., 2002]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Pei et al., 2014a]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Engin and Gur, 2001]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Zwingenberg and Benra, 2006]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Agostinelli et al., 1960]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Aoki, 1984]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Okamura, 1980]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Savillus and Benra, 2006]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Wood et al., 1965]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Parrondo-Gayo et al., 2002]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Benra et al., 2005]</td>
</tr>
<tr>
<td>Radial thrust</td>
<td>●</td>
<td>○</td>
<td>[Agnastoupoulos, 2006]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>○</td>
<td>[De Souza et al., 2008]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>○</td>
<td>[De Souza et al., 2010]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>●</td>
<td>[Majidi, 2005]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>●</td>
<td>[Keays and Meskell, 2006]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>●</td>
<td>[Liu and Tan, 2018]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>●</td>
<td>[Liu and Tan, 2019a]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>●</td>
<td>[Liu and Tan, 2019b]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>●</td>
<td>[NISHI et al., 2012]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Zhou et al., 2013]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Barrio et al., 2008]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[Blanco et al., 2005]</td>
</tr>
<tr>
<td></td>
<td>●</td>
<td>●</td>
<td>[González et al., 2006]</td>
</tr>
<tr>
<td>CFD - k-ε methods</td>
<td>○</td>
<td>●</td>
<td>[Barrio et al., 2011]</td>
</tr>
<tr>
<td></td>
<td>○</td>
<td>●</td>
<td>[Wu et al., 2017]</td>
</tr>
<tr>
<td>CFD - CFD - RNG k-ε</td>
<td>●</td>
<td>●</td>
<td></td>
</tr>
</tbody>
</table>
2.6 Summary

The literature review highlights that harmful phenomena connected to climate change pose new targets in terms of sustainability, environmental conservation and energy management. There are extensive and entwined relationships across multiple frameworks (e.g. social, political, economical, technological) within the energy sector. Current financial measures have been found to be just sufficient to address the intergovernmental agreements, and future predictions are rather uncertain. Suppliers are facing the challenge to provide energy to the network by abandoning fossil fuel sources in favour of renewable energies, although continuously growing population and energy demand are asking for more efficient measures in terms of energy management. This relationship has stimulated new policies frameworks to handle the energy demand from the end-user perspective, by means of demand response and demand-side management measures. Multiple implications are found connecting water and energy in the so-called ‘water-energy nexus’, in particular for wastewater treatment plants, which are intensive energy consumers that service public, residential, commercial and industrial sectors. Within water services, pumping systems are found to be of
the most energy intensive processes.

Studies have shown that methodologies to estimate the energy and environmental footprints are often data sensitive and must consider a multiplicity of inputs and outputs. One of the challenges is to analyse the energy consumption of individual components within industrial plants, to find operational costs. This lack of inclusion involves several issues as numerically assessing the energy consumption of specific devices needs frequently comparisons with real operational duty cycles. Despite the majority of those assessments agreeing that potential savings achievable by reducing the operational costs and increasing energy efficiency of single components or low-size equipment, is clear that energy costs strongly rely on the technologies involved, plant characteristics, and the energy unit cost varying between countries.

A literature review covering the research publications shows the scarce presence of research studies with respect to single-blade wastewater pumps. A large part of those found to be relevant were focused on the effects of the radial thrust due to the asymmetric nature of the impeller. Very few investigations assess the inner fluid mechanics, except for those that investigated the unsteady pressure field induced by the asymmetric shape of the impeller. Moreover, the influence of the span in the impeller-wear plate region is neglected, although in other publications referred to multi-blade centrifugal pumps it is found to be primarily important. Furthermore, numerical simulations on the transient behaviour of a wastewater pump when varying the rotational speed is not addressed. The majority of published numerical studies make use of standard $k$-$\varepsilon$ turbulence model and SST $k$-$\omega$, while other methods, such as RNG or relizable $k$-$\varepsilon$, are rarely used, or not at all. Furthermore, there is a lack of comprehensive studies concerning aspects of the power consumption for different flow regimes.

This study aims to fill gaps in the literature presented in this chapter. Many of the published studies provided an insight of what to expect in terms of flow distribution within the vane, and the resulting radial forces due to the asymmetric shape of the impeller. While there is a scarce presence of works concerning the axial gap in single-blade pumps, the literature review shown that this topic is widely covered for multi-blade centrifugal pumps and turbines. Some authors provided a useful insight about the strategies to numerically assess the leakage flow, in particular, looking at the performance curves and the velocity components at the wear-ring location. On the other hand, the literature review showed little awareness about the performance during transient conditions when a pressure-speed control is applied. Some authors claimed that the performance of a pump coupled with variable speed drives could be calculated just applying similarity curves, although there is no scientific evidence of that. For this reason, a new methodology based on user-defined functions will be applied to the numerical model in order to assess these unexplored subjects.

A novel, more detailed numerical model will be developed to achieve the mentioned goals. A better understanding of the technique will be obtained based on a more adequate modelling and an accurate theoretical description. Strategies to numerically assess the fluid dynamics within the wastewater pumps are described in the next chapter.
Chapter 3

Centrifugal pumps performance

Centrifugal pumps are rotating turbomachines used for the transportation of fluids to raise a specific volume of flow to a specific pressure level [Gulich, 2008]. This family of pumps comprises radial, semi-axial and axial pumps, whose working principles are fundamentally different from that of the first group. A single-stage centrifugal pump is essentially composed of an impeller within a casing (volute) that transfers the angular momentum from a motor shaft to the fluid. The fluid exiting from the impeller is decelerated in the volute and the following diffuser to transform the kinetic energy into pressure energy, therefore increasing the static pressure of the fluid mass. The typical design of such turbomachinery is illustrated in Fig. 3.1.

![Figure 3.1: Submersible sewage pump cross sections, Flygt (Flygt, 2015, Modopump, 2020).](image)

The current chapter aims to present the main aspects of a single-stage, single-entry submersible pump with volute casing. This group very common in water supply, sewage, chemical processing and power plant industrial applications. Conventional sewage pumps are commonly used in the raw sewage treatment and are described as an end-suction, volute-type centrifugal pump with an impeller of either non-clog (Fig. 3.1) or mixed-flow type. Raw sewage liquid typically contains
a large component of complex organics and mineral compounds, such as paper, garbage, rags, sticks, and several other items, with about 25% of the waste as suspension, the remainder as solution. For this reason, the materials of components used depends on the medium to be pumped and the application. These materials may be cast iron, carbon steel, bronze and all kinds of stainless steel as well as plastics, glass and ceramics. The conventional non-clog design typically contains a 2-blades impeller, although some manufacturers have developed a single-blade (or “bladeless”) design with no vane in order to catch trash, which is inherently unbalanced because of the lack of symmetry. Sewage pumps can be “dry” installed horizontally or vertically without being submerged in the pumped liquid. The dry installation is by far the most popular and widely used due to the easier accessibility for maintenance operations, although an additional closed loop cooling system for the electrical motor is required. Moreover, it requires more space to accommodate the base suction elbow, pump-end and its vertical mounted bearing frame, as well as support blocks, shaft, bearings, controls, accessories and security systems [Crawford, 2008]. Wet pit installations are gaining popularity among return activated sludge services, while the submersible type is used in small raw sewage lift stations [Karassik et al., 2001]. Further details for a more in-depth understanding of the basic principles about pumps and more general applications can be found in the literature [Gulich, 2008, Brennen, 2011, Karassik et al., 2001].

### 3.1 Performance characteristics

In centrifugal pumps, the specific work $Y$ is the total useful enthalpy rise $\Delta h_{\text{tot}}$, or total useful energy transmitted to the fluid and is usually measured between the suction and the discharge nozzle. In practice, the hydraulic head $H=Y/g$ is commonly used to measure the total pressure different between two sections of the pump, or the maximum height that a pump can deliver.

$$H = \frac{p_o - p_i}{\gamma} + \frac{V^2_o - V^2_i}{2g} + z_o - z_i \tag{3.1}$$

In eq. 3.1 the head $H$ is measured in metres and is the sum of the differences between static head, velocity head and elevation between the inlet and the outlet of the pump.

$$Y = \Delta h_{\text{tot}} = \frac{p_{o,\text{tot}} - p_{i,\text{tot}}}{\rho} = g \, H \tag{3.2}$$

The specific work in eq. 3.2 can be considered as a specific energy unit, in m$^2$/s$^2$, and the total pressure $p_{\text{tot}}$ is the stagnation pressure plus the gravitational head. Based on these definitions, the useful power transferred to the fluid mass, known as hydraulic power $P_h$, is obtained multiplying the transferred mass flow $m = \rho \times Q$ by the specific work:

$$P_h = \rho g Q H = Q \Delta p_{\text{TOT}} \tag{3.3}$$

Which is equivalent to the product of the volumetric flow rate $Q$ in m$^3$/s by the total pressure difference across the pump $\Delta p_{\text{TOT}}$. The ratio of the hydraulic power and the shaft power $P_s$, where
P_s is determined from the multiplication of the torque T in N and the angular velocity \( \omega \) in rad/s, leads to the global efficiency \( \eta \) as a measure of the effectiveness of transferring the energy from the electric motor to the fluid, and can also be expressed as the product of “component efficiencies”:

\[
\eta = \frac{\rho g Q H}{T \cdot \omega} = \frac{P_{fl}}{P_s} \cdot \frac{\Delta H}{\Delta H_i} \cdot \frac{Q}{Q + Q_L} = \eta_m \cdot \eta_h \cdot \eta_v
\]  

(3.4)

Where:

- \( \eta_m = \frac{P_{fl}}{P_s} \) is the mechanical efficiency, involving that portion of power lost for mechanical power transmission through bearings, seals and frictions;
- \( \eta_h = \frac{\Delta H}{\Delta H_i} \) denotes the hydraulic efficiency that involves the net head \( H_L = H_i - h_L \), expressed as the difference between the ideal head \( H_i \) and the sum of the hydraulic losses \( h_L \) within the main pump passages (e.g. inlet, impeller, diffuser or volute and outlet branch) and are later addressed in section 3.4;
- The volumetric efficiency \( \eta_v = Q/Q_i \), considers the ratio between the effective volumetric flow rate \( Q \), and the ideal flow rate delivered. \( Q_i = Q + Q_L \) and includes leakages \( Q_L \) through the pump cavities.

A more in-depth discussion about losses within the pump channels will be presented in section 3.4.

Centrifugal pumps are designed to operate over a wide range of flow rates, however there are prescribed operational conditions at optimal head \( H_{opt} \) and discharge \( Q_{opt} \) and for a given rotational speed \( \omega \) where they provide the highest efficiency for a given impeller diameter. It follows that such parameters are fundamental to select which impeller and pump type is suitable for a particular application. Pump scaling laws are used to compare performance characteristics among a family of geometrically similar pumps. These laws are constituted by non-dimensional parameters named the head coefficient \( C_H \), flow coefficient \( C_Q \) and power coefficient \( C_W \) as follows:

\[
C_H = \frac{gH}{\omega^2D^2}
\]  

(3.5)

\[
C_Q = \frac{Q}{\omega D^3}
\]  

(3.6)

\[
C_W = \frac{P_s}{\omega D^5}
\]  

(3.7)

It is to be noted that those three parameters are related to the efficiency through the relationship \( \eta = C_HC_Q C_W^{-1} \).

A special case of the similitude theory is how a change in the pump speed affects pump characteristics. By comparing two pumps with the same flow coefficient, the following equations can be defined:

\[
\frac{Q_1}{Q_2} = \frac{\omega_1}{\omega_2}
\]  

(3.8)
\[
\frac{H_1}{H^2} = \frac{\omega_1^2}{\omega_2^2} \tag{3.9}
\]

\[
\frac{P_{1s}}{P_{s2}} = \frac{\omega_1^3}{\omega_2^3} \tag{3.10}
\]

Thus, for a given pump operating at a given flow coefficient, the flow varies linearly with speed, the head with speed squared and the power with the speed cubed.

On the other hand, if the impeller diameter is ignored, the three parameters above can be inter-related by means of the “specific speed” \( N_s \), reported in eq. 3.11, which is of great importance for pump selection and design:

\[
N_s = \omega \sqrt[0.75]{\frac{Q_{opt}}{(gH_{opt})}} \tag{3.11}
\]

where \( \omega \) is in rpm, \( Q_{opt} \) is in \( m^3 \cdot s^{-1} \) and \( H_{opt} \) is in m. The specific speed provides a comparison of hydraulic parameters between impellers with different geometric shapes and features. Specific speed as defined in eq. 3.11 is non-dimensional and varies with the flow coefficient corresponding to peak efficiency only. Each family of pumps have a prescribed range of specific speed associated with their class (e.g. radial, axial, mixed flow, etc), as shown in Fig. 3.2:

![Figure 3.2: Ranges of specific speeds for typical pump geometries [Brennen, 2011].](image)

The classification just presented reflects the fact that some geometries are more efficient at certain specific speeds than others, for instance, axial pumps suits best at high rates and low head, while radial pumps have higher efficiency for low rates and high head. Despite that, the impeller diameter ratio can vary appreciably for a given specific speed. For instance, sewage pumps typically have impeller outlet widths which are twice as large as pumps for non clogging fluids with identical specific speed [Gulich, 2008]. Besides the optimal operating conditions discussed above, pumps usually work away from the design point which is defined as \( Q^* = Q/Q_{opt} \). Performance curves in terms of power are plotted over the range from shut-off (or zero flow at \( Q = 0 \)) to the maximum flow rate they can process. Operational conditions are called “part-load” for \( Q^* < 1 \) and “over-load” at \( Q^* > 1 \).
At a given speed, unique values of head, power and efficiency are ensured at every flow rate. The resulting curves $H = f(Q)$, $P = f(Q)$, $\eta = f(Q)$ are plotted in Fig. 3.3. The head versus flow (H-Q) curve is a steadily falling curve that is measured in tests by throttling a discharge valve, and is reported in Fig. 3.3. The slope varies according the pump type, resulting in a less steep decline for radial flow impellers, as they work well in situations where it is necessary to provide the same head as the flow rate fluctuates (e.g. sprinkle systems). The efficiency curve illustrated in Fig 3.3 steadily increases to a peak (i.e. best efficiency point or BEP), which is unique for a specified impeller, and decreases as the flow rate increases afterwards. Efficiency curves are sensible to the materials used for casting, surface roughness and number of stages. Data sheets from manufacturers are also provided with the required net positive suction head $NPSH_r$, which is the pressure and the kinetic energy (or absolute total pressure), expressed as head. This is the minimum value of the absolute pressure required at the suction port of the pump to avoid cavitation.

Figure 3.3: Typical pump characteristic curves.

The pressure difference divided by the specific weight $\gamma$ defines the $HPSH_r$ as in eq:

$$NPSH_r = \frac{p_i - p_v}{\gamma} + \frac{v_i^2 - v_v^2}{2g} + z_i - z_v$$  \(3.12\)

Cavitation occurs when the pressure at the pump inlet $p_i$ drops below the vapour pressure $p_v$, forming bubbles that are moved to the discharge where they collapse. This phenomenon typically occurs in the proximity of the metallic surface of the rotating blades, exposing them to cyclic stress through multiple implosions at very high temperatures. Cavitation causes irreparable damages to the pump structure, severely reducing the performance. Centrifugal pumps are very susceptible to cavitation at higher flow rates, but it may verify at very low inlet velocities as well. In general, it is more likely to occur when the machine is working off-conditions. These effects must be taken into account when the fluid mechanics of a pump is investigated. Numerical simulations of the
pump behaviour under cavitating conditions are usually accomplished with multiphase flow models, which are not included the present work. However, the operating points have been selected to be away from such unsuitable conditions.

### 3.2 Pumping systems and control

Centrifugal pumps operate within the context of complex hydraulic systems on the need to satisfy the demand, generally in terms of head or flow rate. A pumping system consists of piping, valves, fittings, meters, process equipments and many other liquid-handling conduits and devices besides the pump itself. Each one of these components offer a resistance and generate losses that must be taken into account when selecting one or more pumps. The plot of the sum of those resistances, which are proportional to the square of the flow rate through the system, is called the system head-capacity curve. The additional head to raise the liquid from suction to a higher discharge level plus the pressure difference between suction and discharge conditions that are not proportional to the flow rate are taken into account within the total static head. The system head-capacity curve is constructed considering the major and minor losses within the system to calculate the hydraulic (or head) losses \( h_L \) according to the Eq. 3.13:

\[
\sum \left( K_L + f \frac{L}{D} \right) \frac{v^2}{2g}
\]  

(3.13)

Where \( K_L \) is the minor losses coefficient that takes into account pressure loss through valves and depends on the geometry of the valve and pipe Reynolds number.

![Figure 3.4: Characteristic curves intersection determines the operating point](Al-Khalifah and McMillan, 2012).
3.3 Variable speed control

The coefficients \( f, L \) and \( D \) are the friction factor, length and diameter of the pipe respectively, according to the Darcy-Weisbach equation, and \( v \) is the flow velocity [Munson et al., ]. The friction factor can be estimated from the Moody chart by intersecting the Reynolds number and the relative roughness of the pipe, that depends on the material. For a fixed set of system conditions at constant rotational speed, a centrifugal pump delivers just one flow. This condition is found intersecting the system head-capacity and the H-Q curves, as shown in Fig 3.4, defining the operating point of the pumping system apparatus. This matching point is usually chosen to be at, or near, the best efficiency point and may vary as a consequence of controllable or uncontrollable methods such as opening or closing the valve or the bypass line, ageing of pipes, changes in size or number of pumps or pipes, which alter the shape of the system curves and, in turn, affect the pump flow. Throttle control has been historically used to reduce the flow by increasing the friction in the system. This operation moves the system curve upwards, shifting the operating point towards lower flow rates and setting a new operating point. When a constant flow rate is desired, VFDs are useful to vary the pump speed for an increase, or decrease, of the total system head.

3.3 Variable speed control

The majority of machines employed in industrial processes are driven by electric motors. Pumps belong to those processes responsible for delivering fluids to a desired location, and are equipped with an AC motor, usually of the squirrel cage induction type. The highly conductive material of the laminations, typically made of steel, aluminium or copper, allows the production of a rotating magnetic field through the cage when the stator is crossed with an alternating current, inducing a current in the rotor winding which generates its own magnetic field. The interaction between the stator and the rotor magnetic fields produces a torque on the squirrel cage rotor. This control mechanism can be achieved with a frequency converter that changes the frequency of the alternating current or voltage and this task is performed by variable frequency drives. The AC drive, is assembled by three components: rectifier, DC bus and inverter. The rectifier converts incoming alternating current (AC) to direct current (DC). Since power supplies is three phase, VFDs have typically six rectifiers. Rectifiers built with transistors are preferred to those with diodes or silicon controlled (SCRs). Following the rectifier, the DC bus with capacitors allows the acceptance storage and deliver power through the inverter. The insulated gate bipolar transistor (IGBT) with pulse width modulation (PWM) method is usually the preferred type of inverter as it can be switched on and off several times per second, providing a sine wave current at the desired frequency to the motor.

The input power to the drive system is called total power (measured in kVA) and is the vector sum of the real power (measured in kW), that is the power responsible for the motion, and the reactive power (measured in kVA), and is is proportional to the voltage (in Volts, \( V \)), the current (in Amperes, \( A \)) and the power factor (non-dimensional). Power that charges capacitors or builds magnetic fields in order to cause rotation is called reactive power, which is undesirable as it creates
additional losses and requires larger transformers and wires. The DC bus of VFDs includes capacitors that maintain a high power factor that decrease the reactive power, eliminating the need for additional equipment such as capacitor banks.

The mechanical power at the motor shaft depends on the required torque and the rotational speed. As mentioned earlier, they are controlled by the input voltage, which can be regulated with a frequency converter so that optimal control performance can be achieved to benefit the entire system. The advantage of VFDs is that control systems and the driving motor are located in segregated rooms, which makes maintenance easier compared to mechanical drives. The AC drive motor control diagram is shown in Fig. 3.5a. The electrical supply feeds the electricity to the motor control, which is managed by a user interface for remote monitoring, diagnosis, configuration and control by observing the process response and adjusting the voltage accordingly. When a motor is driven by a frequency converter, the electric motor torque varies with the voltage $V_R$ at the stator and is controlled by the inverter, as shown in Fig. 3.5b. The actual torque at the shaft is found when the motor torque curve intersects the load torque $T_L$ curve, where the load curve consists of friction, inertia and the load itself. Operating conditions relative to higher voltages can be run only for a small amount of time as the motor cooling system is not designed to operate at heavy use continuously.

At specific situations, such as start-up, it can be used 2 times the full load torque, while currents can be 6 to 8 times that experienced at full load. These conditions are undesirable as they can cause failures of other devices across the line. In this context, VFDs are the ideal “soft starters” as they can vary the frequency to limit the power and the current drawn by the motor, providing the lowest inrush compared to other systems (e.g. wye-delta, autotransformer, solid-state starters) [Carrier, 2005]. In addition VFDs using active front-end (AFE) technology allow the control of the power input, increasing the power factor and lowering the harmonics to meet Institute of Electrical and Electronics Engineers (IEEE-519) standards without any need to decrease supply side disturbances.
3.4 Viscous effects

Fluids are *continuum substances* which do not resist to external shear forces, so any force, even the smallest, causes the deformation of a fluid particle. Fluid dynamics is a very broad field, and many books approach fluids theory with different notation.

Centrifugal pumps are turbomachines delivering water at higher pressure and the mechanism can be determined by incompressible single-phase physics for Newtonian fluids. The characterization of turbulence and the architecture of solvers will provide better insights of the numerical modelling process. The conservation laws and a brief description of the turbulence modelling in use are resumed later in this chapter. Detailed analysis of computational techniques can be found in the reported literature [Ferziger and Peric, 2002, Chung, 2002].

Under normal operating conditions, the pressure loss is primarily determined by the thickness of the boundary layer on the suction and pressure sides of the impeller, which strongly influences the wake downstream the blade’s trailing edge. When the pump is running at off-design conditions, the boundary layer and wake thickness increase, because of the separation on the suction side from the leading edge of the blades. On the other hand, there are several mechanisms causing additional losses, especially two: flow separation and three-dimensional secondary flow effects [Horlock and Lakshminarayana, 1973]. Figure 3.6 shows a representation of these effects occurring in multi-vaned centrifugal pumps. At the best efficiency point, the viscous effects are at minimum. However, Fischer et al. [Fisher, 1932] found that even at this operating condition, the wake exists and it extends all the way through the impeller discharge, forming two different regions: a low velocity zone close to the suction surface, and a high velocity (jet) zone in proximity of the blade pressure side.

![Diagram](image1)

(a) Representation of the “jet-wake” interaction.  
(b) Secondary flows within a pump vane.

**Figure 3.6:** Viscous effects within a centrifugal pump [Brennen, 2011].

A representation of the “jet-wake” interaction is shown in Fig. 3.6a. As the operating point goes far from the BEP, the increasing incidence yields to wider wakes creating uneven pressure distributions and slip reduction due to viscous effects, which are function of the flow coefficient $C_Q$ and
the Reynolds number. This slip reduction does not bring any improvement to the Head rise, as the viscous losses are larger than the potential slip reduction. On the other hand, pumps working at high flow regimes, thus at higher Reynolds number, are subjected to other hydraulic losses as consequence of secondary flows and turbulence mixing. The pump vane cross-section may be divided in three different areas: a core region, where the “jet-wake” mixing occurs, a boundary layer region close to the blade surfaces, and an interference region in proximity of the pump casing. The clearance in between allows the fluid to flow back in the impeller region through the gap between the blade and the casing, as shown in Fig. 3.6b, because of the pressure difference between the suction and the pressure sides of the blade. It is known from literature that this effect causes a drop in the performance metrics. Battacharyya et al. [Battacharyya et al., 1993] observed a similar sort of “back-flow” also in the hub region, allowing the flow to re-enter onto the blade passage from downstream. Other disturbances may occur at all flow rates, such the flow separation on the outside and inside of the volute tongue. This incoming reversal flow, when meeting the outgoing stream moved by the impeller rotation, creates local unsteady vortex structures leading to severe vibrations and noise.
Chapter 4

Numerical Methods and setup

4.1 Turbulence modelling

Computational fluid dynamics is based on numerical techniques to study the behaviour of fluid flows, in particular, prediction of bulk quantities (e.g. pressure, velocity) of the system being analysed. In Appendix B, the Navier-Stokes equations are presented as the most general description of fluid dynamics. Equation B.6 states that if $T$ is large enough, $\Phi$ does not depend on time and the time average can be replaced by ensemble average, a process known as Reynolds average. Applying this rule to the Navier-Stokes relationships yields to the RANS equations. Moreover, the presence of Reynolds stresses results in a closure problem since the continuity and the momentum conservation equations, contain more unknown than equations. It would be possible to work around this issue by deriving higher order correlations, but these imply the presence of further unknowns. Therefore, the only way to solve the closure problem is to proceed by using modelling approximations, called turbulence models. This process implies the addition of two (in case of $\kappa - \omega$ models) or seven (for Reynolds stress model RSM) supplementary equations that must be solved through numerical methods, with respective computational effort.

All the quantities are function of the viscosity, which increases because of the turbulent nature of real flows. This suggests that models based on the Boussinesq assumption can be used alternatively and the Reynolds-stress components can be correlated to the mean velocity gradients:

$$- \rho u'_i u'_j = u_i \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu_t \frac{\partial u_k}{\partial x_k} \right) \delta_{ij}$$  \hspace{1cm} (4.1)

Where $k$ is the turbulent kinetic energy:

$$k = \frac{1}{2} u'_i u'_i = \frac{1}{2} \left( u_x' u_x' + u_y' u_y' + u_z' u_z' \right)$$  \hspace{1cm} (4.2)

With $\mu_t$ in Eq. 4.1 is an additional turbulence or eddy viscosity. Dimensional analysis showed that this quantity is derived by the turbulent kinetic energy $k$, the turbulent dissipation rate $\varepsilon$ and a
proportionality constant $C_\mu$, related by eq. 4.3:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$  \hspace{1cm} (4.3)

The presence of kinetic energy diffusion and dissipation rate suggests two additional equations that must be solved to determine the turbulent length scale. This set of equations constitutes the standard $k-\varepsilon$ model [Launder and Spalding, 1974]. This model has been frequently used to determine the fluid dynamics in centrifugal pumps, as pointed out in Chapter 1, and the derivation of the following equations is widely covered in literature [Karassik et al., 2001, Brennen, 2011, Ferziger and Peric, 2002, Chung, 2002, Launder and Spalding, 1974, Guj and Camussi, 2009]:

$$\frac{\partial}{\partial t} \left( \rho k \right) + \frac{\partial}{\partial x_j} \left( \rho u_j k \right) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \sigma_k \frac{\partial k}{\partial x_j} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - T_M$$  \hspace{1cm} (4.4)

$$\frac{\partial}{\partial t} \left( \rho \varepsilon \right) + \frac{\partial}{\partial x_j} \left( \rho u_j \varepsilon \right) = C_{1\varepsilon} \frac{\varepsilon}{k} \left( G_k + C_{3e} G_b \right) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}$$  \hspace{1cm} (4.5)

Where $\sigma_k$ and $\sigma_\varepsilon$ are the Prandtl number for $k$ and $\varepsilon$. Common values for the coefficient above are presented:

$$C_{1\varepsilon} = 1.44 \quad C_{2\varepsilon} = 1.92 \quad \sigma_k = 1.0 \quad \sigma_\varepsilon = 1.3$$  \hspace{1cm} (4.6)

Further improvements were brought to the standard $k-\varepsilon$ for regions of low Reynolds number flow, e.g. near-wall regions, swirling flows, and rapidly strained flows, by an analytical derivation of the Prandtl number and eddy viscosity formulations, called the re-normalization group (RNG), resulting in different constants and additional terms for Eq. 4.4 and Eq. 4.5. The major weakness of these models is modelling the dissipation rate, resulting in poor performances for highly dissipative flows involving rotation, boundary layers under strong adverse pressure gradients, separation and recirculation. These phenomena are particularly important and often manifest in centrifugal pumps. The realizable $k-\varepsilon$ turbulence model amends this issue, accounting for additional constraints with direct physical relevance, thanks to the possibility to prescribe algebraically the turbulent length scales even for high complexity flows [Wilcox, 2008]. It is to be mentioned that $k-\varepsilon$ turbulence models carry some shortcomings should be acknowledged. Any approach with one integral time scale and one integral length scale cannot be universally applicable and accurate to model all the turbulent flow. However, $k-\varepsilon$ models are used in industry because their prediction of the statistical mean properties and the computational effort are found to be acceptable for many applications. In particular, the realizable $k-\varepsilon$ model gives improved predictions than the standard $k-\varepsilon$ model for two reasons. First, it contains a formulation of the parameter $C_\mu$ which is no longer constant but sensible to the mean flow (mean deformation) and the turbulence parameters ($k$ and $\varepsilon$). Second, the determination of the dissipation rate $\varepsilon$ is derived from an exact equation for the transport of the mean-square vorticity fluctuation.

The eddy viscosity models described above have significantly reduced the computational time for a quick and rough approximations of turbulence in numerical simulations. However, their simplicity is also their weakness, being two-equation models. Therefore, they fail to predict flows
4.2 Near-wall modelling

with complex turbulent interactions, such as domains with a high degree of anisotropy, significant streamline curvature, and flows influenced by rotational effects. These anomalies are commonly found when simulating the numerics of centrifugal pumps. To amend this, the exact Reynolds stress transport equation must be solved. The method employed is called second order closure, better known as the Reynolds Stress Model (RSM) [Chou, 1945, Launder and Spalding, 1951], which adds seven partial differential equations, significantly increasing the computational effort:

\[
\frac{\partial}{\partial t}(\rho u_i u'_j) + \frac{\partial}{\partial x_k}(\rho u_k u'_i u'_j) = - \frac{\partial}{\partial x_k} \left[ \rho u'_i u'_k + p' \delta_{ij} + \delta_{ij} u'_j \right] \\
+ \frac{\partial}{\partial x_k} \left[ \mu \frac{\partial}{\partial x_k} (u'_i u'_j) \right] - \rho \left( u'_i u'_j \frac{\partial u_i}{\partial x_k} + u'_i u'_k \frac{\partial u_j}{\partial x_k} \right) \\
+ \rho' \left( \frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i} \right) - 2 \mu \frac{\partial u_i'}{\partial x_k} \frac{\partial u_j'}{\partial x_k} \\
- 2 \rho \Omega_k \left( u'_j u'_m e_{ikm} + u'_i u'_m e_{jkm} \right)
\] (4.7)

Equation 4.7 can be rewritten in a more compact form as:

\[
\text{Local time derivative} + C_{ij} = D_{T,ij} + D_{L,ij} + P_{ij} + \phi_{ij} - \varepsilon_{ij} + F_{ij}
\] (4.8)

Where \( C_{ij} \) is the convection term, \( D_{T,ij} \) is the molecular diffusion, \( P_{ij} \) is the term for stress production, \( \phi_{ij} \) is for the pressure strain, \( \varepsilon_{ij} \) stands for the dissipation rate and \( F_{ij} \) is the force due to system rotation. Despite RSM approach being suitable for complex three-dimensional flows with strong streamline curvature and strong swirl/rotation, it uses more computer memory and computational time and is tougher to achieve a numerically converged solution due to the close coupling of the constituent equations.

It is to be mentioned that \( k-\varepsilon \) turbulence models carry some shortcomings which to be acknowledged. Any approach with one integral time scale and one integral length scale cannot be universally applicable and accurate to model the turbulent flow. However, \( k-\varepsilon \) are used in industry because their prediction of the statistical mean properties and the computational effort are found to be acceptable for many applications. In particular, realizable \( k-\varepsilon \) gives improved predictions than standard \( k-\varepsilon \) for two reasons. First, it contains a formulation of the parameter \( C_{ij} \) which is no longer constant but sensible to the mean flow (mean deformation) and the turbulence parameters (\( k \) and \( \varepsilon \)). Second, the determination of the dissipation rate \( \varepsilon \) is derived from an exact equation for the transport of the mean-square vorticity fluctuation.

4.2 Near-wall modelling

The fidelity of the turbulence models depends on the accuracy of the models for the turbulent transport, pressure-strain correlation and dissipation terms. Also, the models need a highly accurate description of the flow behaviour close to the domain walls, better known as the boundary layer. The domain walls are assumed to be solid boundaries, thus the fluid will have zero velocity
Chapter 4. Numerical Methods and setup

relative to the boundary: a condition known as “no-slip”. The velocity gradient at the wall, and hence the shear stress, is calculated as:

\[ \tau_w = \mu \left( \frac{dv}{dy} \right)_w \]  \hspace{1cm} (4.9)

In a turbulent flow, the velocity profile has at least two different length scales associated with the flow. The first corresponds to the region closer to the wall, the viscous sublayer, with large gradients, dominated by viscous effects. The velocity in the outer region is comparable to that of the mean flow. When Reynolds number is high, the viscous sub-layer becomes very thin such that an huge amount of nodes would be necessary to capture the velocity gradients within the boundary layer region, where from stationary at the wall the flow evolves to be fully turbulent. Typically, a wall function on a logarithmic scale is used to compute the velocity profile, as shown in Fig. 4.1.

In the logarithmic layer, the profile is described by the equation:

\[ u^+ = \frac{v_t}{v_f} = \frac{1}{\kappa} \ln y^+ + C \]  \hspace{1cm} (4.10)

where \( v_t \) is the mean velocity parallel to the wall and \( v_s = (|\tau_w|/\rho)^{1/2} \) is the shear velocity, in which \( \tau_w \) is the shear stress at the wall, \( \kappa \) is the Von Karman constant (\( \kappa = 0.41 \)) and \( C \) is an empirical constant related to the viscous sub layer thickness. Looking at Fig. 4.1, it is possible to define four main regions within the boundary layer based on the values of \( u^+ \) and \( \ln(y^+) \):

- The viscous sublayer \((y^+<5)\), dominated by viscous effects. In the viscous sublayer, velocity fluctuations are induced by the turbulence above the viscous sublayer, with much more dissipation than production of turbulent kinetic energy;
4.2. Near-wall modelling

- The buffer layer, connecting the viscous and logarithmic layer, where dissipation and inertial effects are somewhat balanced;
- The logarithmic layer, where turbulent kinetic energy production and dissipation are nearly equal in this region, and that the diffusion is almost zero;
- The outer layer, beyond the logarithmic layer, beginning approximately at one tenth of the boundary layer thickness.

It is the presence of the wall that gives rise to the turbulent momentum boundary layer, which has the steepest gradients in the inner-most portion of the boundary layer. Prediction of drag, pressure drop due to separation, and heat transfer are critically influenced by the modelling of near-wall region.

When approaching a problem with RANS simulations, considerations of the fidelity level of the resolution of turbulent quantities must be made. This can be accomplished by varying the refinement of the computational grid near the domain boundaries. From the discussion above, the $y^+$ value provides information about how the flow around the boundary layer is resolved. The non-dimensional distance $y^+$ is expressed as:

$$y^+ = \frac{\bar{v} y}{\nu} \tag{4.11}$$

where $y$ is the absolute distance from the wall and $\nu$ is the kinematic viscosity. The $y^+$ value is a non-dimensional distance (based on local cell fluid velocity) from the wall to the first mesh node. For those situations where integrating through the viscous sublayer is necessary (e.g. heat transfer, aerodynamic drag, flows with adverse pressure gradients), a value of $y^+$ approaching 1 is necessary. For less demanding situations, such as cases where wall-bounded effects are secondary or where the flow separates to a dominant shear flow due to geometry such as in the case of a bluff body, values of $y^+$ should be in the range of 30-300. In this range, the viscous and buffer layer are not directly solved, instead, semi-empirical formulas known as “wall functions” are used to bridge the viscosity-affected region between the wall and the fully-turbulent region. In ANSYS Fluent, the lower limit for the log-law is $y^+ \geq 11.225$.

The use of wall-functions obviates the need of a finer mesh resolution, thus requiring lower computational effort. Some turbulence models (e.g. $\omega$-based models, LES) enables the viscosity-affected region to be fully resolved, aka “Low-Reynolds Modelling”. These kind of approaches demand a $y^+ < 1$ everywhere in the computational domain to predict accurate values of the wall shear stress. The adoption of $y^+$ values in $\varepsilon$-based models may result in unbounded errors in wall shear stress and wall heat transfer due to the damping functions inherent within the wall function approach. Therefore, the appropriate refinement of the wall adjacent mesh and the most appropriate wall function is recommended to ensure confidence of the simulation results.

The wall functions approach is suitable for some high Reynolds number flows, especially when integration through the viscous sub-layer has minimum impact on the solution accuracy. This kind of approach works well for wall-bounded problems but fails to predict the development of
flows outside of constant-shear and local equilibrium assumptions. As the flow within sewage centrifugal pumps is highly sheared and frequently subjected to adverse pressure gradients, a non-equilibrium wall function approach is chosen to model the logarithmic layer and have good predictions of the velocity in proximity of the wall, affecting the mean flow. This is based on the two-layer concept to compute reciprocal relations between turbulence production, the turbulence kinetic energy due to the mean velocity gradients and the dissipation rate by non-equilibrium means. The mean velocity sensitized to the pressure gradients $\tilde{U}$ is drawn from eq. 4.12:

$$
\tilde{U} = U - \frac{dp}{dx} \frac{1}{2} \left[ \frac{y_v}{\rho k} \ln \frac{y}{y_v} + \frac{y - y_v}{\rho k} \frac{y_v^2}{\mu} \right]
$$

(4.12)

Where $U$ is the mean flow velocity and $y_v$ is the viscous sublayer thickness, defined as:

$$
y_v \equiv \frac{\mu \gamma^*}{\rho C_{\mu}^{3/4} k^{1/2}}
$$

(4.13)

With $y_v^* = 11.225$. This formulation is uses in cases where the swirl is generated by pressure gradients. In such cases, non-equilibrium wall functions can improve the predictions since they use the law of the wall for mean velocity sensitized to pressure gradients. In order to be able to solve the kinetic energy turbulent equation at the wall-neighboring cells, the following assumptions are made:

$$
\tau_t = \begin{cases} 
0, & y < y_v \\
\tau_w, & y > y_v 
\end{cases}
$$

$k = \begin{cases} 
\left( \frac{y}{y_v} \right)^2 k_p, & y < y_v \\
\frac{k_p}{\gamma^* k^{3/2} C_{\gamma}} y, & y > y_v 
\end{cases}
$

$\epsilon = \begin{cases} 
\frac{2 \kappa k}{y^2}, & y < y_v \\
\frac{k^{3/2}}{C_{\gamma}} y, & y > y_v 
\end{cases}
$

(4.14)

Where $C_{\gamma} = \kappa C_{\mu}^{3/4}$. The cell-averaged production and dissipation of turbulent kinetic energy are calculated from the volume average of $G_k$ and $\epsilon$ of the wall-adjacent cells. This enables the non-equilibrium wall function to take into account, to some extent, the effect of pressure gradients on the distortion of the velocity profiles which are not not accounted for standard wall functions due to the local equilibrium assumption.

Other methodologies for near-wall treatment include scalable wall functions, enhanced wall treatment (EWT-$\epsilon$) and Menter-lechler $\epsilon$-equation (MT-$\epsilon$) equations, although they will not be discussed as the dissertation would be out of the scope of this study. Other near-wall modelling choices include Large Eddy Simulation (LES) approach, although the mesh requirements of this technique make it not suitable for the present assessment. Despite the weaknesses here presented, the realizable $k-\epsilon$ turbulence model with non-equilibrium wall functions was deemed to be the most suitable choice to assess the energy performance of the single-blade wastewater pump.

### 4.3 Solution algorithms

Computational fluid dynamics make use of numerical approaches to solve the Navier-Stokes equations, which in general can be divided into pressure-based and density-based [Shyy and Mittal, 1998]. The density-based algorithms are suitable for compressible flows and problems involving...
high Mach numbers and the continuity, momentum and energy equations are solved simultaneously (i.e. coupled together), with pressure related to density through an equation of state. However, this algorithms become unstable and possess low convergence rate for low Mach number problems, which is common for incompressible flows. The fluid dynamics of incompressible flows can be solved using pressure-based algorithms, although in their set of equations no explicit formulation for the pressure is directly available, thus it must be indirectly specified via the continuity equation. There exist two different formulations to handle this problem: the coupled and the segregated approaches. They mainly differ in the way that the continuity, momentum and (when appropriate) energy equations are solved, as shown in Fig 4.2.

Figure 4.2: Segregated (left) and coupled (right) pressure-based algorithms [Fluent, 2017].

In the coupled solver, while conservation equations are solved simultaneously, additional scalars are solved sequentially (e.g. segregated from one another and from the coupled approach). By looking at the right side of Fig. 4.2, each iteration consists of the following steps:
1. Update the flow properties based on the current or initialized solution;

2. Continuity, momentum and (eventually) energy are solved simultaneously;

3. Mass flux is updated and so are other equations for scalar and turbulent quantities, energy, species and radiation intensity using the updated value of the other variables;

4. A convergence check is done: the algorithm stops if convergence criteria are met, iterating again otherwise.

One of the drawbacks of coupled methods is their high memory requirement to store all the coefficients, which must be repeatedly calculated. In the segregated approach, pressure is determined from a given velocity field and the individual governing equations are solved sequentially. The method, which is shown in the left side of Fig. 4.2, can be outlined as below:

1. Upload the flow properties based on the current or initialized solution;

2. Solve the momentum equation from the pressure and face mass fluxes from point 1;

3. Solve pressure-correction equation using recently updated velocity field and mass face flux;

4. Solve the equations for scalar and turbulent quantities, energy, species and radiation intensity using the updated value of other variables;

5. A convergence check is done: the algorithm stops if convergence criteria are met, iterating again otherwise.

In pressure-based algorithms, pressure and velocities are called primitive variables and there are many methods to find their solution for incompressible flows. The most popular formulation was the Marker and Cell (MAC) proposed by Harlow and Welch, lately extended by Patankar and Spalding with the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) [Harlow and Welch, 1965, Patankar, 2018]. The SIMPLE algorithm obtains the pressure field by enforcing the mass conservation from a relationship between velocity and pressure correction. If the momentum equation does not satisfy the continuity equation, a correction \( J_f' \) is added to the face flux \( J_f^* \) so that the corrected face flux \( J_f \) is:

\[
J_f = J_f^* + J_f'
\]  \hspace{1cm} (4.15)

with:

\[
J_f^* = \hat{J}_f^* + d_f \left( p_{c0}^* - p_{c1}^* \right)
\]  \hspace{1cm} (4.16)

where \( d_f \) is a function of the average of the momentum equation coefficients for the cells on either side of the face \( f \) and \( p_{c0} \) and \( p_{c1} \) are the pressures within the two cells on either side of the face \( f \). The corrected face flux then can be written as:

\[
J_f' = d_f \left( p_{c0}' - p_{c1}' \right)
\]  \hspace{1cm} (4.17)

where \( p' \) is the cell pressure correction.

Besides other methods deriving from SIMPLE scheme (e.g. SIMPLER, SIMPLEC), the Pressure-Implicit...
with Splitting Operators (PISO) is based on a higher approximation degree for the pressure and velocity correction. This method arises from one of the major SIMPLE drawbacks; that is calculations can be longer when velocities and corresponding fluxes do not satisfy the momentum balance after the pressure-correction equation is solved. The PISO scheme performs additional steps in computing the pressure-correction, as shown in Fig. 4.3, using neighbor and skewness corrections.

![Flowchart](Figure 4.3: The PISO scheme [Fluent, 2017].)

Within the skewness correction, the pressure gradient is recalculated and used to update the mass flux correction. This operation works well especially for meshes with a high degree of skewness, producing results at approximately the same number of iterations as more orthogonal grids. The neighbor correction consists in another iteration for the velocity and pressure correction to satisfy continuity and momentum equation more closely. Although this process can extend the computational time for each iteration it reduces the number of iterations required for convergence, especially for transient problems, making it suitable for the simulations accomplished in this work.
4.4 Wastewater pump geometry

The particular pump analysed in this study is the XFP PE-2 150E CB 1.1, designed and manufactured by Sulzer, shown in Fig. 4.4. The design shown is the polished version after the machining process and it consists of three main components: the wear plate, the single-blade impeller and the volute casing. The arrows indicate the path followed by the water flow during pumping operations.

The electric motor which would be installed below the volute casing, not shown in the figure, rotates the impeller about an axis aligned with the inflow direction. The wear plate connects the impeller with the inflow pipe and has an internal diameter of 150 mm and an external diameter of 280 mm.

The inner side of the wear plate presents a spiral groove that reduces friction, separations and wear between the rotating and the stationary elements. The impeller is a single-blade contrablock design, i.e. it is specifically designed to avoid blockage due to the passage of solid objects and slurries and has a maximum diameter of 239 mm with a “ball passage” width of 100 mm. The materials used range from cast iron (EN-GJL-250) to stainless steel 1.4470 (AISI 329), typically flame hardened or ceramic coated. The impeller trailing edge slightly protrudes over the base plate to optimize hydrodynamic performance. The assembly design is a compromise between performance and what is easily machined in the manufacturing process. Maintaining a small gap in this area is important to prevent pressure losses from the pressure side of the blade to the suction side. The double curvature volute has a minimum and maximum radius of 166.91 mm and 223.46 mm respectively,
4.4. Wastewater pump geometry

with a discharge diameter of 150 mm. The electric motor provides a rated power of 9 kW and a nominal speed of 1475 rpm at 50 Hz, with up to 15 starts per hour. The distance \( s_g \) indicated in Fig. 4.4 is relative to the closest point between the impeller blade and the wear plate. According to the information provided by the manufacturer, this distance is set to a maximum to 0.3 mm during assembly and can be adjusted using the mounting bolt system which secures the wear plate to the volute casing. This is to minimize flow leakage which negatively affects performance and also to prevent debris from building up in the gap, blocking the pump.

![Graphs](image)

**Figure 4.5:** Sulzer XFP PE2 150E CB 1.1 experimental data set.
An assessment about variation in the performance metrics due to increased axial clearance is discussed in Chapter 6. Figure 4.5 shows the pump performance curves of the single-blade wastewater pump. The curves have been obtained from the data sheet available at the manufacturer’s website at reference values for the working fluid (water) of 998.3 kg·m$^{-3}$ for the density and 1.005 mm$^2$·s$^{-1}$ for the kinematic viscosity. The head-discharge curve reported Fig. 4.5a shows that the hydrodynamic range of the pump decreases with increasing flow rate, varying from 22 m at 24.6 m$^3$·h$^{-1}$ to 5.4 m at 320 m$^3$·h$^{-1}$. The operational conditions for the highest efficiency (e.g. best efficiency point) is found at 10.88 m and 196.8 m$^3$·h$^{-1}$. At these conditions, the efficiency of the pump, which is shown in Fig 4.5c, is 75 %, while the mechanical power at the shaft, shown in Fig. 4.5d, is 7.68 kW. The mechanical power curve is obtained by multiplying the torque measurements by the angular velocity, rated at 1473 rpm for the current data set. The experimental data here presented was recorded by the manufacturer through in-house testing over a large amount of samples, therefore results as the average over numerous revolutions. Uncertainties in the experimental data are declared to be in agreement with the ISO9906:2012. According to the standard, uncertainties for each variable are: 8% for the head, 10% for the flow rate, 7% for the shaft power (reported as torque in Fig. 4.5b) and 6% for the efficiency.

### 4.4.1 Geometry modelling

The numerical model is built from a computer-aided design (CAD) model of the pump assembly. All the three parts (volute, impeller, wear-plate) were smoothed and rounded off to facilitate meshing. The entire adjustment procedure was accomplished using SOLIDWORKSTM 2016. For the volute, only small modifications were made. Figure. 4.6 shows a comparison between the unmodified and the revised volute geometry. As can be seen, the screw passages and caps were removed from the design to avoid the creation of non essential fluid regions.

![Figure 4.6: Original (left) and modified (right) volute.](image-url)
Major modifications were made on the single-blade impeller, shown in Fig 4.7. The waved cut below the base plate was suppressed and replaced with a squared cut to reduce the number of curves in the impeller design. For the same reason, the base plate was flattened to avoid having faces at different height along the disc. The two cylinders on the spiral blade surface serve the purpose of preventing the build-up of sediments behind the impeller and to allow the shaft clamping. Both the holes were suppressed, to reduce the grid density around those regions. The original impeller required particular effort to refine the mesh around the blade tip because of the presence of sharp corners between the tip and the plate. As a result, the mesh for the original impeller had a higher number of low quality elements (i.e. high skewness), increasing the computational cost by a factor of approximately three compared with the modified trailing edge model. This was assessed by looking at the average time between two successive iterations. A comparison between the original and the modified trailing edge geometry is shown in Fig 4.8.

In this modified geometry, the blade tip was truncated to avoid having cells with poor aspect ratio, resulting in a modified trailing edge on the pressure side. This model resulted in relatively larger thickness of the blade and higher outlet angle with respect to the base plate. The reason for this modification is simplify the design by removing the spikes and the hard edges of the impeller trailing edge for mesh quality purposes, in an attempt to balance performance accuracy with computational cost. Results showing the change in performance metrics between the original and the modified trailing edge are presented in Chapter 5. Substantial modifications were made to the wear-plate, shown in Fig. 4.9. Beside the suppression of the screw holes, the anti-clogging spiral groove on the wear plate was removed and modelled as a flat surface. This feature has a depth 3 mm and may well influence the flow around the impeller.
The axial distance between the impeller and the wear-plate, labelled as $s_g$, was set at 0.3 mm as default. This is the nominal design gap size. For the definition of the fluid domain, the assembly has been imported to SOLIDWORKSTM and intersected with a solid cylinder to fill the cavities within the pump assembly. Once all the redundant bodies were removed, two pipes consisting of cylinders 150 mm in diameter and 600 mm long were attached to the inflow and outflow inlets. The pipe length was chosen to be the shortest in order to reduce the element number outside the rotating region, to ensure that there were not significant changes in the output quantities. However, the pipe length was chosen to be long enough to avoid the boundary conditions affecting the flow. The fluid domain was exported from SOLIDWORKSTM as a part and was successfully imported in the computational software ANSYSTM Workbench for further adjustments. In the geometry modeller of the computational software, the geometry was reviewed and checked for possible importation failures. Existing slivers, spikes, holes, hard edges and sharp angles and interrupted curves were manually corrected. The imported volume was sliced into four different regions (e.g. inlet and outlet pipe, impeller and volute) and renamed conveniently. The final fluid domain is shown in Fig 4.10.
4.4. Wastewater pump geometry

The volute and the outlet pipe were joined as a single body. The impeller region was cut as a cylinder with 255 mm in diameter to allow the creation of two distinct faces with zero thickness between the impeller and the volute. This was necessary to allow the sliding mesh method to be set appropriately. The impeller and the inlet pipe were treated as two separate bodies to allow the relative motion. Finally, the entire domain was set as a fluid zone and transferred to the integrated meshing software.

4.4.2 Meshing strategy

An unstructured grid was applied to the computational domain using a patch conforming method. All faces and boundaries (edges and vertexes) was meshed with tetrahedra, as shown in Fig. 4.11 and Fig. 4.12. CFD was set as physics reference and Fluent was chosen as preferred solver. The reason of this choice is that, while other computational software are good for turbomachinery (e.g., CFX), they focus on multibladed devices. In addition, at the outset, it was not clear if cavitation would be a significant part of this study, and so Fluent, which is superior for simulations involving multiphase flows, was the preferred choice. A more detailed discussion about these software comparisons can be found in the Appendix B. Proximity and curvature was set as sizing function with a minimum element thickness of 0.02 mm, and maximum face and tetrahedra size of 10 mm. All the other parameters were left to default values. The $y^+$ values range from 30 to 300 on the static walls, with an average of 200 on the rotating region. This was verified by plotting the $y^+$ value on the stationary and rotating walls when the convergence criteria were met. This parameter strictly depends on the turbulence model and wall treatment in use. For $k-\varepsilon$ turbulence models, a $y^+$ value
of between 30 and 300 is sufficient to resolve the near-wall region. Other turbulence models re-
quire a much higher refinement at the wall boundaries. As pointed out in section 4.1, the SST $k-\omega$
requires a value of $y^+ \approx 1$ everywhere in the computational domain, thus a $y^+ = 30$ is too large to be
acceptable. As realizable $k-\varepsilon$ is chosen for the current simulations, the near-wall treatment with a
$y^+ = 30$ can be considered resolved for the purpose of URANS equations. For the inlet pipe, a sweep
method was chosen because of the cylindrical nature of the body. It was not possible to apply the
same grid structure to the outlet pipe, as it was connected to the volute to form a single body. This
expedient was required to avoid having the interface acting as a wall. The impeller, volute and out-
let pipe regions were meshed with patch conforming method. The same settings in terms of mesh
density were kept for all the simulated cases.

The grid refinement in the gap region (section A-A in Fig 4.7) is shown in Fig. 4.13 for a gap size of
$s_g=0.4$ mm. With these settings, the different geometry configuration associated with the range of
axial gap clearances from 0.3 mm to 2 mm resulted in a mesh count between 2.6 and 5 million cells.
The following results in chapter 5 are presented with the finest grid to ensure more reliable numer-
ical accuracy. A grid independence study was first conducted to ensure that the mesh size and the
temporal discretization were adequate in terms of numerical accuracy. The three grid refinements,
ranging from 2.6 to 4.6 million cells shown no significant variation (<2%) in the bulk output quan-
tities (e.g. pressure head, torque, efficiency) with increased mesh density. The assessment of the
temporal discretization was conducted with an impeller rotation angle of $1^\circ$, $3^\circ$ and $6^\circ$.
The 3° step showed a 2% deviation in head and power compared to the 1° step, suggesting that 3° is sufficient. A convergence criterion consisting of three sub-criteria is applied to address the enhance confidence in the numerical solutions. The residuals for continuity and momentum drop at
least five orders of magnitude, while variations in global mass flow imbalance and averaged bulk output quantities between successive revolutions are below 1%, as shown in Fig. 4.15. A similar approach was adopted by Melzer et al. [Melzer et al., 2018], suggesting that these considerations offer adequate confidence in the numerical solution. The initial setup for steady simulations was run at constant nominal speed of 1475 rpm with an input velocity of 3.12 m·s\(^{-1}\), corresponding to the flow rate where the pump operates at the highest efficiency. 20,000 iterations were decided to be appropriate to reach the residual convergence criteria of \(10^{-5}\) for continuity, momentum and turbulent quantities. The steady solution was given as input for the unsteady simulations, involving the rotation of the rotating region through the sliding mesh technique, for 20 impeller revolutions.

![Scaled residuals](image1)

*Figure 4.14:* Iterative convergence of scaled residuals

The scaled residuals for four different time steps reflecting the residual convergent criterion are reported in Fig. 4.14a. These results are further compared to a similar trend observed in literature, as shown in Fig. 4.14b.

![Output cycle-averaged ratios](image2)

*Figure 4.15:* Output cycle-averaged ratios at 198 m\(^3\)·h\(^{-1}\): ▲ momentum, □ static pressure.
Figure 4.15 shows the output values of numerical quantities (e.g. pressure, torque) averaged over each revolution. The term $\phi$ in Fig. 4.15 is a non-dimensional variable expressed as the ratio between the outputs at the n-th revolution and the same outputs at the last revolution:

\[
\phi = \frac{p_n}{p_{20}} \tag{4.18}
\]

\[
\phi = \frac{T_n}{T_{20}} \tag{4.19}
\]

where $p_n$ and $T_n$ are the cycle-averaged static pressure at the inlet interface and torque at the n-th revolution and $p_{20}$ and $T_{20}$ are the cycle-averaged static pressure and torque at the last revolution (for $n=20$), respectively.

It is shown that as the unsteady simulations are run, the observed quantities settle asymptotically around the final value, achieving periodic behaviour. It can be seen that this is achieved after only 10 revolutions for the momentum and 13 for the pressure, where the discrepancy between successive cycles is less than $10^{-3}$. A similar behaviour was observed for all the analysed cases, thus 20 revolutions were sufficient to ensure the independence over run time.

### 4.4.3 Solver setup

After the application of the computational grid, the numerical model was imported into the solver to setup the numerical simulations. The CFD software in use is ANSYS FLUENT 19.2.

Three-dimensional RANS equations are solved in a pressure-based formulation to model the incompressible flow. Water at 23°C is chosen as working fluid with a density of 998.2 kg/m$^3$ and dynamic viscosity of $1.003 \times 10^{-3}$ kg·(m·s)$^{-1}$. Turbulence closure is achieved using a realizable $k$-$\epsilon$ model, with non-equilibrium wall functions. Interfaces between impeller and volute and between impeller and inlet pipe are set to allow the faces to rotate relatively. The volute and the outlet pipe are set as stationary frames while the impeller and the inlet pipe are set to rotate at a constant speed of 1475 rpm clockwise with respect to the positive y-axis. For steady simulations (e.g. frozen rotor), the impeller walls are set as rotating with zero relative velocity with respect to the cell zone, to allow the walls and the adjacent cells to move at the same rotational speed. On the other hand, the inlet pipe walls are set to move at 1475 rpm in the opposite direction with respect to the cell zone. This allowed the inlet pipe walls to be stationary while simulating a pre-swirl within the duct. A no-slip condition is applied to all faces to ensure zero velocity at the wall boundary, except for the rotating walls where only tangential velocity is specified as non-zero and equal to the wall rotational speed. The flow velocity is specified at the inlet pipe boundary as a uniform velocity profile with a fixed value according to Table 4.1 with respect to the design flow rate $Q_n$ at the best efficiency point, while the pressure outlet is set to be at atmospheric pressure of 101325 Pa. PISO algorithm is chosen as pressure-velocity coupling scheme. Skewness and neighbor correction weights are kept to a default value of 1 while skewness-neighbor coupling is turned off to avoid
excessive computational times. With PISO algorithm, this technique allows a more accurate adjustment of the face mass flux correction according to the normal pressure correction gradient [Issa, 1986].

Table 4.1: Inlet velocity values.

<table>
<thead>
<tr>
<th>Inlet velocity $u_i$ [m$^3$/s]</th>
<th>Flow rate $Q$ [m$^3$/h]</th>
<th>Relative flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.73</td>
<td>110</td>
<td>55% $Q_n$</td>
</tr>
<tr>
<td>2.19</td>
<td>140</td>
<td>70% $Q_n$</td>
</tr>
<tr>
<td>2.67</td>
<td>170</td>
<td>85% $Q_n$</td>
</tr>
<tr>
<td>3.12</td>
<td>198</td>
<td>100% $Q_n$</td>
</tr>
<tr>
<td>3.6</td>
<td>230</td>
<td>116% $Q_n$</td>
</tr>
<tr>
<td>4</td>
<td>254</td>
<td>130% $Q_n$</td>
</tr>
</tbody>
</table>

Gradients are computed with least squares cell because the accuracy of this method is superior to cell-based gradient when using irregular (skewed and distorted) unstructured meshes. A second order approximation is used for pressure and momentum. Residual convergence criteria are set to $10^{-5}$ for continuity, momentum and turbulent quantities. Static, dynamic and total pressure are monitored computing the area-weighted average at the inlet and outlet faces. Torque is calculated reporting the momentum coefficient of the rotating impeller. Mass flow rate is reported monitoring the flux through the inlet and outlet pipe. The solution is initialized with hybrid initialization method using a number of iterations as default. After that, full multi-grid initialization (FMG) is used to provide better initial solution and faster flow convergence [Fluent, 2017]. The steady simulations are run until residuals convergence is achieved, requiring twenty thousands iterations.

For the unsteady simulations, URANS equations are solved in a first order implicit transient formulation. Impeller, volute and pipes conditions are set to mesh motion to enable the sliding mesh technique. The position increment is set to an angular displacement of $\Delta \varphi = 3^\circ$ by using a time step size $\Delta t = 3.39 \times 10^{-4}$ s. The time step is calculated according to eq.:

$$\Delta t = \frac{t_n}{360 \Delta \varphi}$$  \hspace{1cm} (4.20)

Where $t_n$ is the time in seconds to complete a single revolution. At the design speed of 1475 rpm, the time to complete one revolution is equal to $4.07 \times 10^{-2}$ s, calculated as:

$$t_n = \frac{1}{\omega}$$  \hspace{1cm} (4.21)

Where $\omega$ is the angular velocity in revolutions per seconds. At this design speed, almost 25 revolutions are required to simulate a flow time of 1 s. A different time step size is calculated when the rotational speed is varied, as shown in chapter 7.

In a different study, an inlet pressure is specified as an in-flow boundary condition at the inlet pipe. Pressure is varied from 70 kPa to 160 kPa with 15 kPa increments, leading to eight different simulation cases. Furthermore, the rotational speed was varied from 80% to 130% of the nominal value with 10% increments. In the transient parametric simulations, user-defined functions (UDFs) were
used to simulate a change in the inlet pressure boundary conditions and change in the rotational speed each time step. The inputs given to the boundary conditions for the transient simulations in chapter 5 are shown in Table 4.2.

<table>
<thead>
<tr>
<th>N. of rotations</th>
<th>N. of time steps</th>
<th>( \Delta P ) [Pa]</th>
<th>( \Delta N ) [rpm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>600</td>
<td>161.667</td>
<td>0.9833</td>
</tr>
<tr>
<td>15</td>
<td>1800</td>
<td>53.889</td>
<td>0.32778</td>
</tr>
<tr>
<td>25</td>
<td>3000</td>
<td>32.3333</td>
<td>0.19667</td>
</tr>
</tbody>
</table>

The variables \( \Delta P \) and \( \Delta N \) are the incremental pressure and rotational speed that are summed to the previous values of pressure and rotational speed each time step, respectively. A more detailed insight of the UDF codes used to simulate dynamic boundary conditions is reported in Appendix A. Because of the heavy computational effort required, simulations were run on the Ireland Centre for High-End computing (ICHEC) cluster. A scalability study accomplished with the cluster’s supercomputer (Kay), having 40 cores per node, revealed that the speed up varies approximately linearly between one and two nodes, but falls off rapidly. Therefore, two nodes were chosen to run each simulation, with a total wall-clock time of 72 hours.
Chapter 5

Transient solution at constant rotational speed

In order to analyse the flow field in relation to the impeller position, a suitable reference frame is defined and shown in Fig. 5.1.

![Figure 5.1: Computational model reference frame.](image-url)
Chapter 5. Transient solution at constant rotational speed

The stationary reference frame is marked as $z,x$, with $\varphi$ indicating the rotational position of the impeller with respect to the $z$-axis. The angle $\varphi$ increases in the clockwise direction and it is set to $0^\circ$ when the blade tip (trailing edge) is aligned with the volute tongue.

5.1 Internal flow results

A comparison of the pressure field at different flow rates for the same impeller position ($\varphi = 0^\circ$) at the mid-cross plane ($y=0$) is shown in Fig. 5.2. Since the static pressure boundary condition was set to be 0 Pa at the outlet, the pressure values are considered with respect to the reference pressure (101326 Pa). According to the contour plots shown in Fig. 5.2, the pressure and the suction side are consistent with the gross behaviour that would be expected around the impeller: an over-pressure zone is shown on the side of the blade facing the rotation direction.

![Figure 5.2: Instantaneous static pressure distribution at $y = 0$ (mid-section plane) for $\varphi = 0^\circ$: a) $Q = 110 \, \text{m}^3\cdot\text{h}^{-1}$, b) $Q = 198 \, \text{m}^3\cdot\text{h}^{-1}$, c) $Q = 254 \, \text{m}^3\cdot\text{h}^{-1}$.](image)

This is relative to the mechanism of energy exchange (i.e. momentum) between the rotating structure (i.e. the impeller) and the fluid, which is pumped towards the discharge pipe. On the other hand, the suction side is clearly visible on the inner part of the blade relative to the rotation axis, corresponding to an under-pressure zone. This is relative to the mechanism of drawing the fluid
from the inlet pipe to the impeller vane. The variation in the pressure field magnitude is consistent with the change in the inlet boundary conditions: high flow rates correspond to lower static pressure differences in the volute region (i.e. lower head), while low flow rates involve higher static pressure differences (i.e. higher head). It is seen that the static pressure distribution in the volute region also gradually decreases in the rotation direction. From a numerical perspective, this is caused by pressure rise due to the impeller motion, which has greater impact on the pressure field compared to the inlet boundary conditions. In a physical pump installation, the flow rate is set by the pressure difference and the pressure loss in the pipe network. More realistically, the pressure rise due to the impeller motion is reduced more in a high flow rate situation (i.e. when the pressure loss in the pipe network is low). At the blade tip, where the high pressure and the low pressure fluid merge, three distinct phenomena are observed. By looking at Fig. 5.2 and 5.3, an under-pressure zone is found between the high pressure and the low pressure zones.

Also, two pressure areas form around the impeller tip sharp edges, indicating the presence of two stagnation points. While the appearance of these phenomena is less visible at lower flow rates, as in Fig. 5.2a, it becomes pronounced with increased flow rates, as shown in Fig. 5.3b. The reason behind this effect can be found by looking at the absolute velocity vectors in the volute region. Figure 5.4 shows the absolute velocity vectors at the mid-cross plane \((y = 0)\) for three different flow rates and at the same impeller position of \(\phi = 0^\circ\). It can be seen that as the flow rate increases, the velocity vectors become more aligned with the \(z\) axis, i.e. towards the discharging pipe. It is also observed that at higher flow rates, a small part of flow tends to recirculate from the suction side of the impeller to the outlet, joining the incoming flow from the pressure side directed to the discharging pipe. The meeting of these flows yields a low velocity zone around the blade tip, justifying the localised increased pressure zones observed in Fig. 5.3a and 5.3b. It is also shown that the under-pressure front is bent more towards the tongue as the flow rate increases.
Chapter 5. Transient solution at constant rotational speed

Figure 5.4: Velocity vector field at $y=0$ (mid-section plane) for: a) $\varphi = 0^\circ$ at $Q = 110 \text{ m}^3\cdot\text{h}^{-1}$, b) $\varphi = 0^\circ$ at $Q = 198 \text{ m}^3\cdot\text{h}^{-1}$, c) $\varphi = 0^\circ$ at $Q = 254 \text{ m}^3\cdot\text{h}^{-1}$, d) $\varphi = 246^\circ$ at $Q = 198 \text{ m}^3\cdot\text{h}^{-1}$

Figure 5.5: Velocity vectors with flow rate $\varphi$ [Nishi and Fukutomi, 2014a]
5.1. Internal flow results

This may be due to a higher localised flow velocity between the impeller blade, which rotates at constant speed, and the volute walls. Besides the possible explanations regarding the appearance of this phenomenon, it is difficult to claim it as physically real. While there is no localized pressure or velocity experimental data for this pump, the results discussed above are consistent with the overall pump behaviour reported in the literature (see [Nishi and Fukutomi, 2014a], [Blanco et al., 2005]), as shown in Fig. 5.5. By looking at the velocity vectors Fig. 5.4, a stagnation point at the volute walls is observed for all the examined cases. As expected from published data, the stagnation point moves along the volute tongue, relocating further from the discharge pipe as the flow rate increases. At low flow rates, Fig. 5.4a, a low velocity zone appears in the discharge pipe with higher velocity vectors, simulating re-entrant flow in the volute region. At the best efficiency flow rate, Fig. 5.4b, the stagnation point is located after the volute tongue in the same direction as the rotating impeller. At higher flow rates, Fig. 5.4c, the stagnation point moves even further, with higher velocity flow moving from right to left around the tongue. While the stagnation point at lower flow rates is formed at the tongue because of the separation of the outgoing and re-entrant flows, at higher rates is due to the recirculation of flow from the suction side towards the outlet. This effect may be caused by the increased separations that often results when the pump is working far from the design point.

![Figure 5.6: Instantaneous static pressure distribution at y = 0 (mid-section plane) at Q = 198 m³·h⁻¹ for: a) ϕ = 0°, b) ϕ = 90°, c) ϕ = 180°, d) ϕ = 270°.](image-url)
It is also observed that the position of the stagnation point changes with while the impeller rotates. Figure 5.4d shows that for $\varphi = 246^\circ$, the stagnation point is located exactly at the tongue. While this configuration corresponds to the instantaneous peak pressure for the best efficiency flow conditions, it is also observed that the amount of flow re-entering the rotating region is the minimum for all the examined cases.

![Figure 5.4d: Static pressure fluctuation at the inlet face over a complete revolution for Q = 198 m$^3$·h$^{-1}$.](image)

The instantaneous pressure distribution at midspan for four impeller positions is shown in Fig. 5.6 for a flow rate of 198 m$^3$·h$^{-1}$. The instantaneous static pressure at the inlet interface, which is used to calculate the developed head, is also shown in Fig. 5.7 and relative to the impeller positions shown in Fig. 5.6. It is observed that the outlet pressure increases for $27^\circ < \varphi < 246^\circ$, decreasing for all the other values of $\varphi$. It is clear that the pressure are at minimum values when the blade tip approaches the volute tongue (i.e. look at static pressure values in figure below from 345$^\circ$ to 75$^\circ$), indicating lower head. The contour plot in Fig. 5.6a clearly shows an over-pressure distribution above the pressure side of the impeller. In order to highlight the pressure gradients in the rotating
region, the pressure contour of Fig. 5.6a is plotted in a local scale, so it does not take into account the pressure values at the domain boundaries. The pressure fluctuations in Fig. 5.7 show similar trend to what would be expected in a centrifugal pump. A comparison with published data, showing a trend similar to Fig. 5.7, is reported in Fig. 5.8. The authors found that the radial thrust of the impeller is lower in proximity of $\varphi = 0^\circ$, yielding to lower values of the outlet pressure.

![Pump Head](image)

**Figure 5.8:** Transient pump head over a complete revolution for different boundary conditions [Melzer et al., 2018].

On the other hand, the radial thrust is large when the blade tip is passing at the opposite side of the volute tongue. This is consistent with the higher magnitude of the static pressure fluctuations observed in Fig. 5.7. The outcomes for the lateral forces are analysed more in detail in chapter 8. A time-dependent non-dimensional pressure coefficient $C_p$ is used to evaluate the instantaneous static pressure at the inlet interface with the flow rate for a complete impeller revolution, and is shown in Fig. 5.9. The pressure coefficient is a function of time and it is defined as follows:

$$C_p = \frac{p(t) - p_{\text{ref}}}{\frac{1}{2} \rho u_2^2}$$

(5.1)

Where $p(t)$ is the instantaneous static pressure at the inlet, $p_{\text{ref}}$ is the static pressure averaged over the full revolution at $Q = 198 \text{ m}^3\cdot\text{h}^{-1}$ and $u_2$ is the impeller circumferential tip speed. As the flow rate increases, the static pressure drops for all the reported impeller positions. As the head losses
are proportional to the square of the flow velocity as discussed in Chapter 4, the back-pressure at the inlet face is decreased as the inlet velocity is increased.

![Figure 5.9: Pressure coefficient at the inlet face over a complete revolution](image)

The developed head is then calculated by taking the time-average of the values shown in Fig. 5.9. It is observed that the impeller position at the peak pressure changes with the flow rate. Higher values of pressure coefficient are found for \( \varphi = 234 \) at \( Q = 110 \text{ m}^3\cdot\text{h}^{-1} \), \( \varphi = 246^\circ \) at \( Q = 198 \text{ m}^3\cdot\text{h}^{-1} \) and \( \varphi = 270^\circ \) at \( Q = 254 \text{ m}^3\cdot\text{h}^{-1} \). Similar behaviour is also found when the pressure coefficient values are at its lowest value, although the angle shift between the minimums is approximately the half. In fact, while \( \Delta \varphi = 12^\circ \) between the higher pressure values for \( 110 \text{ m}^3\cdot\text{h}^{-1} \) and \( 198 \text{ m}^3\cdot\text{h}^{-1} \), \( \Delta \varphi = 6^\circ \) when the lowest values are found between the same flow rates. The same offset was found by taking the cycle-average of the static pressure each revolution for the same impeller position, thus it appears to be as a periodic effect of the pump operation. To give a possible explanation of the peak shift observed in Fig. 5.9, the vorticity in the pump vane for \( \varphi = 270^\circ \) is shown at different flow rates (\( Q = 110 \text{ m}^3\cdot\text{h}^{-1} \), \( Q = 170 \text{ m}^3\cdot\text{h}^{-1} \), \( Q = 198 \text{ m}^3\cdot\text{h}^{-1} \), \( Q = 254 \text{ m}^3\cdot\text{h}^{-1} \)) in Fig. 5.10. Figure 5.10 shows the internal flow development at similar impeller position \( (\varphi = 270^\circ) \), which is approximately when the instantaneous static pressure is at its peak value. The distribution of the vorticity in the rotating region with Q-criterion is also shown. It is observed that as the flow rate increases, the wake structure grows downstream of the impeller trailing edge. This is manifested through a growth in the wake length and size of the toroidal structure, surrounding the impeller geometry up to the
5.1. Internal flow results

Volute tongue, at $Q = 254 \text{ m}^3\cdot\text{h}^{-1}$.

![Figure 5.10](image)

**Figure 5.10:** Distribution of the vortex core (Q-criterion) in the pump vane for $\varphi = 270^\circ$ at: a) $Q = 110 \text{ m}^3\cdot\text{h}^{-1}$, b) $Q = 170 \text{ m}^3\cdot\text{h}^{-1}$, c) $Q = 198 \text{ m}^3\cdot\text{h}^{-1}$, d) $Q = 254 \text{ m}^3\cdot\text{h}^{-1}$.

At low flow rates, separation from the suction side may occur, while at high flow rates, the separation occurs along the pressure side. On the other hand, this could be due to the increasing flow rotational intensity in the pump vane, which is greater as the flow rate increases. This hypothesis is confirmed by looking at the velocity streamlines also shown in Fig. 5.10. While at low flow rates the streamlines are aligned in the sense of the impeller rotation, their path becomes more chaotic as the inlet velocity increases above the design flow rate. It is clearly visible that the velocity streamlines revolve around the torus-shaped structure of the vortex core. Around the wear side of the impeller blade a flat turbulent structure is also visible whose amplitude in the radial direction decreases as the flow rate is increased. This turbulent zone is concentric to the impeller’s eye on the volute side. It may be that at lower inlet velocities, there is a small portion of flow that is less likely to be dragged downstream in the impeller vane, and that recirculates in proximity of the volute walls generating a flat vortex structure. At higher flow rates, this phenomenon is less pronounced, because of the leakage flow from pressure side to suction side.
5.2 Original and modified trailing edge comparison

In order to assess whether the numerical model was able to replicate the pump energy performance, the simulated head-discharge curve, the torque, and the efficiency, are compared with experimental data in Fig. 5.11. A comparison between the original and modified trailing edge is carried out. The gap size is 0.3 mm (i.e. the nominal design gap) and the flow rates simulated correspond to those in Fig. 5.2. The head-discharge curve is shown in Fig. 5.11a as the product of the inlet pressure divided by the specific weight of the water. As the mass flow rate was given as an input boundary condition, the dynamic pressure was the same at the inlet and outlet of the domain, because these sections have the same pipe diameter and operate under the assumption that the flow is uniform across both boundaries. It follows that only the cycle-average static pressure difference was considered to compute the total head. The data obtained using the original trailing edge shows an over-prediction for all the performance metrics shown in Fig. 5.2. In particular, the pressure head in Fig. 5.11a is approximately 4.5 m greater than the modified trailing edge at low flow rates, with an overestimation of almost 2.7 m above the experimental data at 110 m$^3$·h$^{-1}$. While the discrepancy between the numerical results were unchanged at higher flow rates, the distinction between the original and the experimental data was found to be 1.3 m at 254 m$^3$·h$^{-1}$.

The numerical data representing the modified trailing edge shows an underestimation of the head-discharge in comparison with the experimental data. In particular, it shows a discrepancy of 2.6 m at 110 m$^3$·h$^{-1}$ and 3.4 m at 254 m$^3$·h$^{-1}$. This difference is ascribed to the trailing edge shape, consisting of a lower blade angle and higher width for the modified trailing edge with respect to the original. It is known that this arrangement tends to increase the deviation between the flow and the blade angle, leading to a lower pressure head [Gulich, 2008]. It is likely that at low inlet velocities, the relative velocity between the blade and the flow is small, thus viscous shear forces are less important in such scenarios.

On the other hand, the original trailing edge shows higher efficiency at high flow rates. It is observed that at lower flow rates and up to 198 m$^3$·h$^{-1}$, the experimental data and the numerical solution for the original trailing edge result in almost the same efficiency, shown in Fig. 5.11c. While both the experimental data and the numerical solution for the modified trailing edge show a peak in the efficiency at a flow rate of 198 m$^3$·h$^{-1}$, this operational condition does not correspond to the best efficiency point for the original trailing edge, which is instead shifted to 230 m$^3$·h$^{-1}$. For greater inlet velocities, the numerical efficiency starts to decrease slowly. It is worth noting that the efficiency is calculated as the ratio between the hydraulic power and the mechanical power at the rotor shaft. The latter is the product of the momentum (i.e. torque), shown in Fig. 5.11b and the rotational speed of the impeller. Despite the simulated outcomes of the the original and the modified trailing edge showing a constant decrease in the head with the flow rate, two different trends are observed in the torque data. While the numerical torque constantly increases with the inlet velocity up to 198 m$^3$·h$^{-1}$ for both geometries. A discrepancy of 6.5 N·m is observed at 110 m$^3$·h$^{-1}$, increasing to 12.9 N·m at 254 m$^3$·h$^{-1}$ with respect to the experimental data. Conversely, the data for the original trailing edge increases with the flow rate, which is consistent with the experimental data, showing a discrepancy of 7.8 N·m at 110 m$^3$·h$^{-1}$ and 9.3 N·m at 254 m$^3$·h$^{-1}$. 
Figure 5.11: Variation of pump performance metrics with volumetric flow rate for different impeller geometries: a) Variation of Head with flow rate, b) Variation of Torque with flow rate, c) Variation of efficiency with flow rate (□ Experimental data (from Fig. 4.5); ◇ Modified trailing edge; ○ Original Trailing edge).
It is worth mentioning that the numerical software calculates the momentum as the sum of the pressure forces and the viscous forces acting on the blade. As the pressure shows a constant decreasing trend in Fig. 5.11a, the different behaviour observed in the momentum is likely to be ascribable to the different turbulent viscosity on the impeller walls. This behaviour is reflected in the head reported in Fig. 5.11, which is underestimated for the modified case. This could be due to an underestimation of the viscous losses within the pump channel, which can be attributed to the use of statistical turbulence model. Another explanation could be that the numerical model does not capture adequately how the angular momentum of the impeller is converted into fluid pressure. This is clearly visible at higher flow rates. The discretization scheme could also be too coarse and some of the information concerning the turbulence models could be lost. This could happen because the assumption of isotropic turbulent flow implies that the viscous forces are over-predicted, so viscous dissipation is underestimated. This could be the cause of excessive turbulent viscosity ratio that was observed in other numerical simulations concerning the original trailing edge for flow rates higher than 254 m$^3$/h, which have not been reported due to solution divergence. It is concluded that the numerical model with the modified trailing edge geometry with velocity-inlet and pressure-outlet boundary conditions is sufficient to capture the performance trend and hence the relative change in performance for different operating conditions.

### 5.3 Performance metrics with pressure-inlet pressure-outlet boundary conditions

Figure 5.12 shows the replicated energy performance obtained with pressure-inlet and pressure-outlet boundary conditions. The pressure-inlet is set to 0 Pa, while the static pressure outlet varies from 55 kPa to 160 kPa with 15 kPa intervals. With these settings, the mass flow rate fluctuates with the rotating impeller, thus Fig. 5.12 shows the time-averaged values of the flow rate over the last revolution. It is observed that the trend of the performance metrics is consistent between similar impeller geometries. The pressure head in Figure 5.12a is obtained by taking the total pressure difference between the outlet and the inlet faces. The trends for the original and modified trailing edge with the pressure-pressure boundary condition follow the same pattern as for the velocity inlet and pressure-outlet boundary conditions is sufficient to capture the performance trend and hence the relative change in performance for different operating conditions.

It is worth mentioning that for the modified trailing edge it was chosen to simulate a wider spectrum of operating conditions, most of the simulations concerning the original trailing edge geometry had convergence issues. This was observed when the outlet static pressure was set lower than 70 kPa, at 115 kPa, and at 85 kPa, which is close to the best efficiency point. This complication is observed in the lack of corresponding data points in the torque (Fig 5.12b) and efficiency (Fig. 5.12c) curves for the original trailing edge. The numerical model of modified trailing edge shows superior convergence stability with respect to the original geometry. The modified geometry therefore provided the possibility to analyse a much wider set of operational conditions but also to perform simulations in a shorter computational time.
5.3. Performance metrics with pressure-inlet pressure-outlet boundary conditions

Figure 5.12: Variation of pump performance metrics with volumetric flow rate for different impeller geometries: a) Variation of Head with flow rate, b) Variation of Torque with flow rate, c) Variation of efficiency with flow rate (□ Experimental data; △ Original trailing edge, velocity inlet; ▽ Original trailing edge, pressure inlet; ⊲ Modified Trailing edge, velocity inlet; ♦ Modified Trailing edge, pressure inlet).
While the original trailing edge geometry needed 4 s per iteration on average, requiring approximately 12 hours to accomplish 20 revolutions with unsteady simulations, the modified geometry demanded half of that time. Because of the high number of simulations involved in this study and the limited amount of CPU hours available on the external cluster, the lower computational effort observed in the modified geometry was crucial to determine which model could be run for a wider spectrum of operating conditions. The performance metrics compared in Fig. 5.12 do not show particular advantages of using one type boundary condition with respect to another in terms of numerical accuracy. Both the original and the modified geometry models show similar numerical discrepancy when compared to experimental data. Moreover, the lower convergence stability makes the original geometry unsuitable when dynamic boundary conditions are used, as they require the computational model to be stable over a wider range of operational conditions. For these reasons, only the modified geometry was considered for the gap and transient flow analyses described in the following chapters.

5.4 Summary

The initial stages of this study aimed to generate a high fidelity computational model that was able to replicate the performance metrics of a given pump with an existing experimental data set. The modified trailing edge model has demonstrated higher stability, being able to meet convergence criteria with lower computational efforts. The numerical simulations accomplished with pressure-inlet and pressure-outlet boundary conditions have shown the weaknesses of the computational model of the original geometry, and solution residuals were observed to diverge over a wide range of operating conditions with respect to the modified geometry. Moreover, the numerical solution in terms of energy performance shown good agreement with experimental data, resulting in similar performance for both models in terms of accuracy. For such reasons, the modified geometry was preferred to the original to carry out additional numerical simulations that are shown in the following chapters.

The internal fluid mechanics of the modified geometry was studied in detail and the main outcomes are here summarized:

- The contour plots of the static pressure within the pump vane were found to be in good agreement with the theory: over-pressure and under-pressure zones were identified around the impeller blade, on the pressure side and on the suction side, respectively. The presence of stagnation points at the blade tip and the change in features at different flow rates were found to be in agreement with the numerical simulations and the experimental data found in literature.

- A stagnation point in proximity of the volute walls was found to be exactly at the tongue when the pump was running at the best efficiency point and the impeller position at $\varphi = 246^\circ$ caused stronger static pressure fluctuations at the outlet over the period. In such conditions,
the amount of re-entrant flow was the smaller. The displacement of the stagnation point, along the walls in the rotation direction, found good agreement with the literature.

- The maximum and the minimum values of the pressure fluctuations were found at $\varphi = 246^\circ$ and $\varphi = 27^\circ$ respectively, when the pump was running at $198 \text{ m}^3 \cdot \text{h}^{-1}$. The pressure coefficient was found to decrease as the flow rate is increased, which is consistent with the numerical and the experimental energy performance metrics.

- Streamlines of the flow field within the rotating regions were plotted for different flow rates. The distorted curvature with respect to a plane orthogonal to the impeller rotation axis indicated the presence of increasing turbulence with the flow rate. The distribution of the vorticity with the Q-criterion revealed the presence of a toroidal vortex at the core of the revolving streamlines, downstream of the impeller trailing edge.
Chapter 6

Unsteady force excitation

As discussed in chapter 2 the majority of the investigations concerning wastewater pumps focus on the flow-induced oscillations caused by the unsteady pressure field around the rotating impeller. The asymmetric geometry of single-blade pumps causes unbalanced hydrodynamic forces resulting in periodic excitation.

Figure 6.1: Non-dimensional hydrodynamic forces in stationary coordinates at $s_g = 0.3$ mm and $\omega = 1475$ rpm.
This will occur even at the design point and stronger loads will be observed at off-design operating conditions at the design point [Agostinelli et al., 1960, Aoki, 1984, Okamura, 1980]. This effect causes damages to pump components (e.g. bearings) due to the transmitted vibrations across the pump structure. In order to further validate the three-dimensional computational model, an assessment of the unsteady lateral forces is presented. Moreover, a methodology to observe the spectral components to compare the non-dimensional forces amplitude with the gap size and with the flow rate is presented further in this chapter. The hydrodynamic forces acting on the impeller at pressure inlet 115 kPa during a single revolution along the three axes are shown in Fig. 6.1.

### 6.1 Unsteady forces

The forces $F_x^*$, $F_y^*$ and $F_z^*$ are normalized by a reference force according to eq. 6.1 [Pei et al., 2012a]:

$$F_{ref} = \rho \frac{u_2^2}{2} b_2 r_2$$

where $u_2$ is the circumferential velocity of the trailing edge, $b_2$ is the blade height and $r_2$ is the impeller radius at the blade tip, respectively. The straight arrow in Fig. 6.1 indicates the value of the hydrodynamic forces when the impeller trailing edge is aligned with the tongue at $\varphi = 0^\circ$.

![Figure 6.2: Orbit curves with cycle-to-cycle oscillations from experimental data [Benra, 2006].](image)
The force vector indicating the trailing edge position rotates in the counterclockwise direction and the reported data is relative to a single impeller revolution. In the current simulations, the impeller is constrained to pure rotation. It is observed that the instantaneous hydrodynamic forces acting on the impeller surface would cause the rotor to oscillate along the three axes. The intensity of the forces is strongly asymmetrical, resulting from the uneven distribution of the flow field around the impeller blade. It should be remembered that the forces could be decomposed as the vector sum of the pressure forces and the viscous forces acting on the impeller surface. It is worth noting that the trajectory of the points shown in Fig. 6.1 is smooth, and no oscillations are observed. Therefore, the force frequency is the rotation frequency. A similar behaviour is already observed in the literature in one-way coupled numerical techniques [Benra, 2006, Pei et al., 2012b], where the deflection of the structure due to pressure fluctuations is neglected. In real applications, pumps are sensitive to the pressure fluctuations of the fluid which cause the impeller to vibrate. This effect would cause several oscillations in the orbit curve of Fig. 6.1, implying that the system is subjected to cycle-to-cycle variations [Agostinelli et al., 1960, Aoki, 1984, Okamura, 1980], as shown in Fig. 6.2.

Figure 6.3: Non-dimensional hydrodynamic forces with gap size at design flow rate and $\omega = 1475$ rpm (- $s_g = 0.3$ mm; - $s_g = 0.6$ mm; - $s_g = 1.15$ mm; - $s_g = 2$ mm).
The modelling choices adopted in the current study assume that all the cycles are the same because of periodic behaviour. As a result of having a periodic signal, it is possible to observe the pump behaviour by looking to a single cycle. Figure 6.3 shows the non-dimensional hydrodynamic forces with the gap size plotted in a two-dimensional Cartesian coordinate system. The arrow indicates the position of the impeller when the trailing edge is aligned with the volute tongue and follows the rotation in the counter-clockwise direction. A negative value of the forces indicates a rotation of the rotor towards the negative part of the axes as shown in Fig. 5.1. In order to investigate the behaviour of the forces in the pump vane, only the radial components are considered. The axial component along the y-axes is not reported. The reason is that the component along the y-axis is about four times smaller in amplitude with respect to the components along the x and z axes. This is due to the smaller back-pressure drop along the impeller axis. Moreover, the impeller shaft is typically more resistant to loads along the axis (traction and compression), than bending, although that depends on the axial thrust bearings.
By looking at Fig. 6.3 it is observed that the force distribution along the x-axis and z-axis resembles an elliptic shape which is asymmetric with respect to the major and minor axes. At the starting position for $\phi = 0^\circ$, the blade tip is aligned with the tongue. The force values decrease with increasing gap size, being maximum at $s_g = 0.3$ mm and minimum at $s_g = 2$ mm. However, this behaviour changes at $\phi = 180^\circ$, where the maximum of the absolute force value is higher at $s_g = 2$ mm and minimum at $s_g = 1.15$ mm. It is also observed that along the z-axis, the force is about the same for all the examined cases. This behaviour denotes that the presence of the gap has a major impact on the impeller deflection along the radial direction when the developed pressure is at the maximum and minimum peaks within the cycle. The non-dimensional forces with rotational speed for an outlet static pressure of 115 kPa and a gap size $s_g = 0.3$ mm is shown in Fig. 6.4. It is observed that as the rotational speed increases, the force amplitude increases in the radial direction along the x-axis and is reduced along the z-axis. In particular, when the position of the impeller trailing edge is at $\phi = 180^\circ$, the force $F_x^*$ is more negative at lower rotational speed, and the absolute value is higher. On the other hand, the force $F_z^*$ is reduced with increasing the rotational speed.

Figure 6.5: Non-dimensional hydrodynamic forces with flow rate at $s_g = 0.3$ mm and $\omega = 1475$ rpm ($\bullet$ 17 m$^3$·h$^{-1}$; $\square$ 113.7 m$^3$·h$^{-1}$; $\triangle$ 239.3 m$^3$·h$^{-1}$).
Chapter 6. Unsteady force excitation

The elliptic shape of the orbit curves become more flattened as the rotational speed is increased, with decreased eccentricity. In particular, a peak is observed towards the positive x-axes for \( z = 0 \), where a peak in amplitude of the force \( F_x^* \) is observed. It should be remembered that those coordinates correspond to a trailing edge position close to the volute tongue, which is known to be the cause of strong pressure fluctuations [Majidi, 2005]. Figure 6.5 shows the non-dimensional hydrodynamic forces at different flow rates for constant values of the rotational speed \( (\omega = 1475 \, \text{rpm}) \) and gap size \( (s_g = 0.3 \, \text{mm}) \). It is observed that as the flow rate increases, the force decreases in the positive direction of the x-axes and increases towards negative values. It could be that when the impeller blade approaches the tongue, the intensity of static pressure fluctuations is reduced.

![Figure 6.5: Non-dimensional hydrodynamic forces at different flow rates for constant values of the rotational speed and gap size.](image)

**Figure 6.5:** Non-dimensional hydrodynamic forces at different flow rates for constant values of the rotational speed and gap size.

**Figure 6.6:** Cross-correlation of the hydrodynamic forces and time delay shown as a phase for \( p_m = 115 \, \text{kPa}, \, s_g = 0.3 \, \text{mm} \) and \( \omega = 1475 \, \text{rpm} \): a) Time series, b) Phase \( (-\bigcirc F_x^*; -\bigdiamond F_z^*) \).
This could be caused by the reduced normal stress acting on the impeller surface due to the lower pressure at the outlet, which is typical at high flow rates. At lower flow rates, the orbit curve presents more eccentricity with minor amplitude of the force $F_x'$ when the trailing edge is in proximity of the volute tongue. On the other hand, at higher rates the rotor orbit is nearly circular, indicating similar values of the pressure distribution around the impeller surface during the cycle. This is evident by looking at the gradual increase in amplitude of both the radial forces ($F_x'$, $F_z'$) in the second and third quadrant of Fig. 6.5. For the operating conditions at the design point, for constant rotational speed and a gap size of $s_g = 0.3$ mm, the relative phase between the non-dimensional forces $F_x'$ and $F_z'$ is shown in Fig. 6.6. Figure 6.6a shows the forces as a time series based on the trailing edge position relative to the volute tongue, while Figure 6.6b shows the cross-correlation between the time series. It is observed that both $F_x'$ and $F_z'$ show approximately the same maximum amplitude within the cycle, although a shift in the maximum and minimum peaks is evident. The trend of the time lag shown as phase shift in Fig. 6.6b is found applying a cross-correlation of the two discrete time series relative to the radial forces, normalized with respect to the reference force. The time lag between the two forces is found at the absolute maximum of the cross-correlated amplitude in Fig. 6.6b, for $\Delta \varphi = 84^\circ$.

### 6.2 Spectral components

Looking at the peak shapes in Fig. 6.6a, it is clear that both time series are sinusoidal, but the trend for the non-dimensional $F_x$ shows a sharper peak at $\varphi = 282^\circ$. The non-dimensional force $F_z'$ shows a dominant phase shift due to a different physical phenomenon. The instantaneous radial forces $F_x$, $F_y$ and $F_z$ at the design point are further analysed to investigate the periodic components within the numerical results, and they are shown in Fig. 6.7. The data relative to the time series is fitted with a series of sinusoids using a poly-harmonic function according to Eq. 6.2 [Keays and Meskell, 2006]:

$$F = \sum_{k=1}^{M} [A_k \sin (kT) + B_k \cos (kT)] \quad (6.2)$$

where $F$ is the force, $T = \frac{2\pi}{n_{rev} t_r}$ is the period where $n_{rev}$ is the number of revolutions and $t_r$ time steps per revolution, $M$ is number of harmonics used. $A_k$ and $B_k$ are matrices obtained using a Moore and Penrose pseudo-inverse method which yields a least squares fit for an overdetermined set of equations. In order to analyse the spectral component of the time series, the polyharmonic fit is chosen over the Fast Fourier Transformation (FFT) because it provides the spectrum of a periodic signal for shorter periods. Figure 6.7 shows the fitted multi-sinusoidal of the non-dimensional forces of the last two revolutions. It is observed that all three forces show periodicity with different sinusoidal behaviour. In particular, the non-dimensional force $F_z'$ in Fig. 6.7c is purely sinusoidal. While the comparison with the non-dimensional force $F_x'$ was already presented in Fig. 6.6a, the trend for $F_y'$ is shown in Fig. 6.7b. It is observed that the non-dimensional axial force is periodic but less sinusoidal, so it is expected to have a stronger amplitude at higher harmonic order. Figure 6.8 shows the amplitude of the non-dimensional forces with increasing harmonic order (i.e. the
spectrum) for the operating conditions at the design point. It is observed that for the fundamental harmonic component the non-dimensional forces $F_x^*$ and $F_z^*$ show similar amplitude, which is about three times higher than the magnitude of the non-dimensional force $F_y^*$. As expected, the non-dimensional force $F_y^*$ shows stronger amplitude at higher order harmonic with respect to $F_x^*$ and $F_z^*$. Figure 6.9 shows the spectrum of the non-dimensional forces with the gap size. It is observed that at the first harmonic, the non-dimensional forces $F_x^*$ and $F_z^*$ have approximately the same amplitude, which is four times higher than the non-dimensional force $F_y^*$.

![Graph a) showing $F_x^*$ vs. $n_r$](image-a)

![Graph b) showing $F_y^*$ vs. $n_r$](image-b)

![Graph c) showing $F_z^*$ vs. $n_r$](image-c)

**Figure 6.7**: Polyharmonic fit of the forces time series with number of revolutions (○ Numerical data; ● Fitted multi-sinusoidal).

The amplitude of the non-dimensional force $F_y^*$ is insensitive to the gap size and decreases with increasing harmonic orders. These characteristics are observed for all the examined gaps, and the trend is consistent to what is observed at the design point in Fig. 6.8. At higher harmonic orders, it
is observed that the amplitude decreases with the gap size. In particular, for $s_g = 0.6$ mm, the amplitude of the non-dimensional force $F_{z^*}$ at the $5^{th}$ harmonic is lower than the $6^{th}$ and $7^{th}$ harmonic. The same trend is observed for the non-dimensional force $F_{x^*}$ at the $5^{th}$ harmonic for $s_g = 1.15$ mm and at the $4^{th}$ harmonic for $s_g = 2$ mm, and for the non-dimensional force $F_{z^*}$ at the $4^{th}$ and $5^{th}$ harmonic for $s_g = 1.15$ mm.

Figure 6.10 shows the spectrum of the non-dimensional forces at design gap size $s_g = 0.3$ mm with

![Figure 6.8: Spectra of the the non-dimensional forces at the design point: a) $F_{x^*}$, b) $F_{y^*}$, c) $F_{z^*}$.](image)

the flow rate. It is observed that the amplitude of higher order harmonic is different with respect to the design point. In particular, it is found that when the flow rate is $Q = 110 \text{ m}^3\cdot\text{h}^{-1}$, the amplitude of the non-dimensional force $F_{x^*}$ is the highest for the $3^{rd}$, $4^{th}$, $5^{th}$ and $6^{th}$ harmonic. Similarly, when $Q = 110 \text{ m}^3\cdot\text{h}^{-1}$, the non-dimensional force $F_{y^*}$ is the highest at the $7^{th}$, $8^{th}$ and $9^{th}$ harmonic.
Figure 6.9: Spectra of the non-dimensional forces with gap size: a) - c) 0.3 mm; d) - f) 0.6 mm; g) - i) 1.15 mm; j) - l) 2 mm ($F_x$, $F_y$, $F_z$).
Figure 6.10: Spectra of the non-dimensional forces with flow rate: a) - c) $Q = 110 \text{ m}^3 \cdot \text{h}^{-1}$; d) - f) $Q = 198 \text{ m}^3 \cdot \text{h}^{-1}$; g) - i) $Q = 254 \text{ m}^3 \cdot \text{h}^{-1}$ ($F_x, F_y, F_z$).
6.3 Summary

This chapter discussed the behaviour of the forces of the wastewater pump under different operating conditions considering the variation of the gap size, rotational speed and flow rate. The most relevant findings of this assessment are:

- A three-dimensional plot of the forces shown that uneven pressure fluctuations in the vane cause an asymmetric distribution of the forces around the impeller surface. The absence of local oscillations is explained by the nature of the one-way coupling method used in this study, which does not include the effect of impeller deflection.

- A two-dimensional plot of the forces along the x and z axes in response to increased gap size shows a progressive reduction in the amplitude of the force $F_x$ along the positive side of the x-axis as the gap is increased. The cause could be due to the reduction of the outlet pressure fluctuations in the vane due to leakage around the blade in the axial direction. No significant effects were found regarding the magnitude of the force $F_z$ with increasing gap size.

- The radial forces $F_x$ and $F_z$ were plotted for different rotational speeds showing that as the rotational speed increases, the eccentricity of the force curves is reduced with an over-extension along the major axis in the x direction caused by a larger amplitude of the force $F_x$. This could be caused by the increased pressure fluctuations due to the blade passage in the vicinity of the tongue, which is consistent with the data in literature.

- An assessment of the radial forces with the flow rate showed that as the flow rate increases, the force curves becomes nearly circular, with a reduction in the $F_x$ amplitude and an increase in the amplitude for $F_z$. This is likely due to a reduction in the pressure fluctuations because of the lower pressure head.

- A cross-correlation of the data relative to $F_x$ and $F_z$ at the design point allowed the calculation of the phase delay between the two forces. This was found to be equal to $\Delta \varphi = 84^\circ$;

- The numerical data was fitted with a poly-harmonic function to plot the spectrum of the non-dimensional forces at different harmonic orders. The multi-sinusoidal fit was chosen instead of the FFT because of improved resolution for periodic signals and shorter period.

The assessment reported in this chapter shows that the behaviour of the unsteady non-dimensional forces at off-design conditions is consistent with data presented in literature. Therefore, it offers further confidence in the three-dimensional numerical model of the wastewater pump.
In order to assess the behaviour of the wastewater pump due to impeller-wear plate gap variation, the numerical model was run for six prescribed mass flow rates, from 110 m$^3$.h$^{-1}$ to 254 m$^3$.h$^{-1}$. The rotational speed was set at the nominal design speed of 1475 rpm. The simulations were run for axial gap values from 0.3 mm to 0.6 mm with increments of 0.1 mm and from 0.6 mm to 2 mm with increments of 0.275 mm. To avoid showing an excess of redundant data, only selected data points are presented in the current discussion. The selected values here shown for the axial gap $B$ range from 0.3 mm to 1.15 mm. For each gap size, six flow rates are examined ranging from 110 m$^3$.h$^{-1}$ to 254 m$^3$.h$^{-1}$. For the current analysis, a total of 30 operating conditions (5 gap sizes by six flow rates) are simulated. Unsteady simulations were run for up to 20 impeller revolutions to achieve periodic behaviour of the pump model.

7.1 Effect of the gap on performance metrics

Figure 7.1 shows the simulated performance compared with the experimental data for different gap size. The experimental head-discharge, torque and efficiency curves have been provided by the manufacturer following in-house testing on the hydraulic machine, according to the standard ISO9906:2012. The values from the numerical simulations are averaged over the final revolution. Fig. 7.1a shows the head-discharge curve. At a gap of 0.3 mm (i.e. the design value) the head-discharge variation obtained from simulations approaches the experimental data with an under prediction of approximately 1.7 m at low flow rates and 3 m at high flow rates. As would be expected, as the gap is increased, the developed head decreases at each flow rate. This is manifested as a drop of the back pressure at the inlet interface, which is caused by leakage around the tip of the impeller. Another interpretation of the performance reduction may be that the numerical model does not capture adequately how the angular momentum is converted into pressure. The same phenomenon has already been observed in the performance metrics reported in Fig. 5.11 and in literature [De Souza et al., 2006], as shown in Fig. 7.2. Nevertheless, the fluid dynamics in the mentioned publication was not addressed by a thorough narrative.
Chapter 7. Effect of the axial gap on pump performance at off-design conditions

Figure 7.1: Variation of pump performance metrics with volumetric flow rate for different gaps: a) Pressure Head-Flow rate, b) Average torque about impeller rotation axis, c) Hydraulic efficiency (□ Experimental data; ○ 0.3 mm; ▲ 0.4 mm; △ 0.5 mm; ▽ 0.6 mm; ▲ 1.15 mm).

The hydraulic efficiency, shown in Fig. 7.1c, is specified as the ratio of the difference of hydraulic power between inlet and outlet (i.e. effectively, the stagnation pressure increase) to the shaft.
7.1. Effect of the gap on performance metrics

![Variation of pump performance metrics with experimental data [De Souza et al., 2006].](image)

**Figure 7.2:** Variation of pump performance metrics with experimental data [De Souza et al., 2006].
power applied to the impeller. It should be noted that the experimental value is based on the electricity consumption of the motor, and so embeds a simple estimate of electrical and mechanical losses. On the other hand, the efficiency obtained in the simulations calculates the hydraulic power input directly for the torque acting on the impeller. The effect of the gap observed in the numerical data is to move the maximum efficiency seen in Fig. 7.1c from 198 m$^3$/h at 0.3 mm to 170 m$^3$/h at 1.15 mm. For the gaps in between, the maximum is at 230 m$^3$/h. Fig. 7.1b reports the average torque for different gaps compared with experimental data. Interestingly, the largest changes in the momentum occur for flow rates higher than the best efficiency point. Conversely, the maximum torque at 110 m$^3$/h is shown for 1.15 mm gap. This is in contrast to what is observed for the pressure head, where the developed head constantly decreases as the gap is increased.

![Figure 7.3: Variation of the pressure coefficient with the gap size at 198 m$^3$/h.](image)

It should be remembered that the torque is calculated as the area-weighted average of the pressure and viscous forces acting on the rotating body surface. Therefore, it represents the energy exchange between the rotating walls and the fluid. Nevertheless, the decrease of the torque observed at high flow rates is perhaps due to an under-prediction of the viscous forces. Figure 7.3 shows the variation of the pressure coefficient as defined in eqn. 5.1 with the gap size. The reference pressure chosen is the average pressure over the last revolution at 0.3 mm and 198 m$^3$/h. As expected, the pressure identified by the pressure coefficients decrease as the gap is increased,
7.1. Effect of the gap on performance metrics

which is in agreement with the trends observed for the head-discharge curve in Fig. 7.1. This behaviour is similar to what is observed in Fig. 5.9, where the impeller position that provides the peak pressure is shifted as the flow rate increases.

7.1.1 Sensitivity of performance to gap size

Fig. 7.4a shows the variation of the pump efficiency at the same flow rate with increasing gap size. The efficiency is higher when the pump operates at flow rates around the best efficiency point, and is lower at higher and lower pressure head, as would be expected. However, for gaps greater than 0.5 mm the estimated flow rate for best efficiency point is $170 \text{ m}^3\cdot\text{h}^{-1}$, rather than $198 \text{ m}^3\cdot\text{h}^{-1}$ as found for the nominal design gap of 0.3 mm. As can be seen in Fig. 7.4b, the efficiency drops significantly with increased gap. At a flow rate of $Q=230 \text{ m}^3\cdot\text{h}^{-1}$ the efficiency is reduced by 13.5 percentage points for a gap size $s_g=1.15 \text{ mm}$ compared with the nominal design configuration.

![Figure 7.4](image_url)

**Figure 7.4:** Variation of efficiency with gap size for a range of flow rates: a) Variation of efficiency, $\eta$, with gap size, b) Normalised efficiency with normalised gap ($\square 110 \text{ m}^3\cdot\text{h}^{-1}$; ○ $140 \text{ m}^3\cdot\text{h}^{-1}$; □ $170 \text{ m}^3\cdot\text{h}^{-1}$; ◆ $198 \text{ m}^3\cdot\text{h}^{-1}$; ◇ $230 \text{ m}^3\cdot\text{h}^{-1}$; ▲ $254 \text{ m}^3\cdot\text{h}^{-1}$).

Generally, the efficiency data for all flow rates exhibit a downward trend with increased gap size. At
a flow rate of 230 m$^3$·h$^{-1}$ the data exhibits an increase in efficiency around a gap size of 0.5 mm and 0.6 mm. It is tempting to simply claim this as a numerical artifice. However, the deviation from the trend occurs in two independent simulations ($s_g=0.5$ mm and $s_g=0.6$ mm) each with an independently generated mesh and furthermore, at the largest gap size ($s_g=1.15$ mm), the behaviour at this flow rate returns to the expected trend. While this behaviour may well be due to a numerical artefact, it could also be due to a physical phenomenon. As the gap is increased at a specific flow rate, the local flow around the gap will be prone to shear layer instability, analogous to vortex shedding. In other turbomachinery, this can give rise to so-called non-synchronous vibration [Clark et al., 2013]. While the time and spatial scale of this phenomenon would be too small to be fully captured with the current model resolution, such behaviour even if not fully modelled would act as a blockage, reducing leakage flow and increasing apparent efficiency. Clearly, further work is needed to investigate this anomaly, but the overall trend in Fig. 7.4a is clear.

![Figure 7.5: Sensitivity of efficiency to gap size: a) Variation of non-dimensional sensitivity $K$ with flow rate, b) Variation of efficiency sensitivity factor $K^*$ with flow rate.](image)

While the efficiency is already a non-dimensional quantity, to facilitate comparison with the nominal design configuration, the efficiency can be normalised relative to a reference configuration.
Fig. 7.5b shows the normalised efficiency expressed as the ratio of the efficiency $\eta$ presented in Fig. 7.5a to a reference efficiency $\eta_{\text{ref}}$ chosen as $s_g=0.3$ mm.

Similarly, the gap $s_g$ is normalised with respect to a reference gap length $s_{\text{ref}}=0.3$ mm. It is seen that as the flow rate and the gap size increase, the efficiency ratio drops to 0.91 at 140 m$^3\cdot$h$^{-1}$ and to 0.82 at 254 m$^3\cdot$h$^{-1}$ when the gap is 1.15 mm.

The efficiency ratio drop follows a linear downward trend for each flow rate. The sensitivity of the efficiency ratio to an increase in gap size can be captured as a sensitivity factor $K$, defined as:

$$K(Q) = \left( \frac{\partial \frac{\eta}{\eta_{\text{ref}}}}{\partial \frac{s_g}{s_{\text{ref}}}} \right)$$

(7.1)

![Graph showing variation of mechanical power consumption with gap size for various flow rates.]

The non-dimensional parameter $K$ defined in Eqn. 7.1 is obtained by linear regression of a straight line to the normalised quantities shown in Fig. 7.5b for constant $Q$. As a result, they are a functions
of the flow rate. The coefficient of determination, $r^2$, is greater than 99% for all flow rates except for 230 m$^3$·h$^{-1}$, where it drops to 90% due to the behaviour previously noted around $s_g/s_{ref}$=2. The trend for $K$ is plotted in Fig. 7.5a, signifying that the efficiency becomes more sensitive for higher flow rates. It is seen that $K$ increases up to $Q$=140 m$^3$·h$^{-1}$ and decreases at higher rates. It is also possible to define a dimensional sensitivity factor $K^*$ as:

$$K^*(Q) = K(Q) \cdot \frac{\eta_{ref}}{s_{ref}} = \left(\frac{\partial \eta}{\partial s_g}\right)$$  \hspace{1cm} (7.2)

The parameter $K^*$, defined in Eqn. 7.2 and shown in Fig. 7.5b, has units %·mm$^{-1}$ and quantifies the sensitivity of efficiency to increased gap length in millimetres for each flow rate.

![Figure 7.5a: Variation of non-dimensional sensitivity factor $\tilde{K}$ with flow rate.](image-a)

![Figure 7.5b: Variation of sensitivity factor $\tilde{K}^*$ with flow rate.](image-b)

**Figure 7.5:** Sensitivity of mechanical power consumed $P_m$ to change in the gap size $s_g$: a) Variation of non-dimensional sensitivity factor $\tilde{K}$ with flow rate, b) Variation of sensitivity factor $\tilde{K}^*$ with flow rate.

The sensitivity is always negative, indicating that increasing gap size will always cause a degradation in pump performance. The variation of power consumed with increased gap size is more complicated. Fig. 7.6a shows the mechanical power consumed at each flow rate and gap size. In
order to compare flow rates, the power can be normalized by the power consumed at the nominal design gap ($s_g=0.3$ mm) for each flow rate. This is plotted against the normalized gap size in Fig. 7.6b. As before, the data follows a linear trend for each flow rate. A sensitivity factor $\tilde{K}$ and $\tilde{K}^*$ can be defined:

$$\tilde{K}(Q) = \left( \frac{\partial P_m}{\partial s_g} \frac{s_{g\text{ref}}}{s_{g}} \right)$$

(7.3)

$$\tilde{K}^*(Q) = \tilde{K}(Q) \cdot \frac{P_{\text{ref}}}{s_{g\text{ref}}} = \left( \frac{\partial P_m}{\partial s_g} \right)$$

(7.4)

The parameter $\tilde{K}$ is non-dimensional while $\tilde{K}^*$ has units kW·mm$^{-1}$ indicating the change in power requirement due to increased gap at each flow rate. The variation of $\tilde{K}$ and $\tilde{K}^*$ are shown in Fig 7.7. Except for the lowest flow rate, both $\tilde{K}$ and $\tilde{K}^*$ are negative, indicating a drop in power required to sustain the flow rate as gap size increases. This is due to a drop in viscous loss between the blade and the wear plate. At a flow rate of 254 m$^3$·h$^{-1}$, the power consumption at $s_g = 1.15$ mm drops by 0.58 kW. However, it is worth noting that this reduction in power, which at first may seem advantageous, is achieved with a drop in output pressure head due to leakage, and so there is a drop in efficiency. In practice, this will mean that the flow-rate will be reduced to meet the head requirement, necessitating the pump to run for longer and hence draw more energy.

### 7.2 Assessment of the flow velocity through the gap

In an attempt to understand better the impact of the gap on the pump performance, the cylindrical velocity components, time averaged over a single revolution, are reported in Fig. 7.8 and Fig. 7.9 for a range of gaps and flow rates. The data is taken along a line perpendicular to section A-A in Fig. 4.7. The section location is fixed relative to the impeller. The width of the blade at section A-A is 7.1 mm. The velocities are normalised with respect to the inlet velocity $u_i$ and the location, $s$, in the gap, is normalized by the gap size, $s_g$. The net mass flow in the axial direction, in Fig. 7.8a, must be zero for $s/s_g=0$ and $s/s_g=1$ because of the solid boundaries.

The radial velocity shown in Fig. 7.8b is the most meaningful to understand the mechanism of leakage. The negative value of the normalised radial velocity indicates that the flow particles along $s_g$ move from the pressure to the suction side of the impeller blade (i.e. from outlet towards inlet), causing back-flow. Therefore, it represent the leakage across the axial gap. The absolute value of the radial velocity $u_r$ decreases as the gap is increased. However, the reduction of the absolute velocity is small if compared with the increased mass flow due to the larger passage, thus leakage flow increases proportionally with the gap. Conversely, the axial velocity increases with the gap size, as shown in Fig. 7.8a. A Lagrangian assessment of the fluid flow would show a particle of fluid being swept upwards as the impeller blade advances, through the gap and then downwards as the impeller passes. The tangential velocity in Fig. 7.8c is insensitive to the gap size.
Chapter 7. Effect of the axial gap on pump performance at off-design conditions

Figure 7.8: Variation of velocity components in the gap region for different gaps at 198 m$^3$·h$^{-1}$: a) Axial velocity, b) Radial velocity, c) Tangential velocity, d) Absolute velocity (···△··· 0.3 mm; ···○··· 0.4 mm; ···−··· 0.5 mm; ···□··· 0.6 mm; ·····−··· 1.15 mm).
The collapse in the core region of the tangential velocity pattern might be suggesting a computational artefact. However, the higher number of cells due to increased gap size would rather indicate a physical feature. Everywhere in the gap, the tangential velocity magnitude is lower than the rotational speed of the impeller wall. The velocity drops linearly to zero at the wear-plate stationary wall, where a no slip condition is imposed. The velocity distribution is linear because of the viscous forces causing shear in the boundary layer region, while in the core of the passage there is little influence with increasing the axial clearance.

Figure 7.9: Variation of velocity components in the gap region for different gaps and flow rates: a) Axial velocity, b) Radial velocity, c) Tangential velocity, d) Absolute velocity

The tangential velocity component (Fig. 7.9c) is insensitive to either gap size or flow rate. It is proposed that the tangential component is determined by a combination of the rotational speed and the flow rate. The trend in Fig. 7.9b shows three linear sections. One might be concerned that the
central region with almost constant velocity is a consequence of mesh design. However, this has been investigated and there is no evidence that this is the case. Moreover, each gap size is simulated using a different mesh size, both in terms of cell count and distribution through the gap. The torque associated with shear stress at the impeller depends on the spatial gradient of the tangential velocity at s=0. The fact that the normalised tangential velocity collapses well in Fig. 7.8c implies that the shear stress will scale inversely with both gap size (s_g) and flow rate (u_i), which is consistent with the linear trend of the power consumption with gap size and the linear trend of the power sensitivity factor  \( \tilde{K} \). It is also observed that the axial and the radial components have little influence on the absolute velocity, except to increase the amplitude in the core region of the gap. Similar trends have been observed when the flow rate is varied with the gap size, as shown in Fig. 7.9. The non-zero axial velocities (Fig.7.9a) are due to the the fact that the location of s_g is fixed to the impeller. It is observed that the axial velocity increases with the flow rate and the gap size. On the other hand, the radial velocity (Fig. 7.9b) is dominated mainly by the flow rate. Note that although the normalised radial velocity is comparable for all gap sizes, the radial volumetric flow rate (i.e. the leakage) will be larger for bigger gaps and will vary linearly with gap size. This explains the linear reduction in efficiency with gap size. Curiously, at this location, the radial velocity, which is more negative for higher flow rates, indicates higher leakage. This is counter-intuitive as high flow rates are associated with low pressure difference across the impeller, and hence one might expect lower leakage. On average this will be the case over the entire length of the impeller, but close to the trailing edge, the flow will experience large vortex structures, which will be dominated by inertial effects. The tangential velocity reported in Fig. 7.9c shows little influence from the inlet conditions and similar behaviour of that seen in Fig. 7.8. By looking at the absolute velocity component in Fig. 7.9d, the velocity in the gap is mainly affected by the rotational speed than the flow rate, suggesting that the flow dynamics in the gap is mainly affected by the rotational speed of the impeller. The average magnitude of the flow velocity through the gap is lower at 110 m^3·h^-1, and this is caused by the reduction of the radial velocity at lower flow rates. This phenomenon tends to reduce the amount of leakage when operating at lower head, but this advantage is realised solely for the smallest gap size. The increased leakage by means of an increase of the radial velocity component due to the gap at high flow rates could be reduced by adjusting the rotational speed of the pump. In this context, the use of variable frequency drives becomes of the utmost importance as an increase of the axial clearance cannot be directly measured. This is especially true when accessing the pump is arduous (e.g. wet well installation). In all the other cases, maintenance to keep the gap within design values in order to ensure maximum pump reliability is highly recommended.

### 7.3 Summary

The modified geometry was used to assess the effect of the axial gap on the energy consumption through three-dimensional numerical simulations. It was found that:
7.3. Summary

- Pump performance are found to be degraded as the gap size increases. The pressure head constantly decreases with the gap size and the flow rate, up to 3 m at 254 m$^3$.h$^{-1}$. The under-prediction of the viscous forces in the torque at 1.15 mm caused a shift in the maximum efficiency from 198 m$^3$.h$^{-1}$ at 0.3 mm to 170 m$^3$.h$^{-1}$ at 1.15 mm and 230 m$^3$.h$^{-1}$ for the gaps in between;

- Parametric analysis were carried out to assess the loss in energy performance (e.g. power, efficiency) with the gap size. The pump performance is found to be degraded by increasing the gap size, with the largest drop in efficiency reported as 13.5%. While it is true that for a given flow rate the power consumption is reduced, this is associated with a decrease in the pressure head.

- Non-dimensional sensitivity factors were created to assess the loss in efficiency and power consumption in relation to gap size. The linear correlation allowed sensitivity factors to be quantified, as this will allow the results to be applied to similar pumps or different sizes. The coefficients $\tilde{K}$ and $\tilde{K}^*$ quantify a loss of 13.5%·mm$^{-1}$ and 0.58 kW·mm$^{-1}$ for the efficiency and the power consumption, respectively;

- An assessment of the velocity components revealed that the mechanism of leakage is mainly affected by the axial velocity, representing the motion from the pressure to the suction side. While the magnitude of the flow velocity is mainly driven by the tangential component (i.e. the rotational speed of the impeller), the absolute velocity was found to increase with the flow rate, resulting in more leakage.

The literature review in chapter 2 identified the lack of frequent effective maintenance to avoid energy losses due to increased axial clearance. Although the effect of gap size on energy performance of a wastewater pump is smaller than the reported benefit of variable speed drives, it is potentially significant. Therefore, it is concluded that as well as the capital intensive strategies to improve total pump efficiency, such as variable speed drives, it is important to implement a proactive maintenance protocol throughout the lifetime of the installation [Caruso and Meskell, 2021].
Chapter 8

Transient performance with variable speed drives at off-design conditions

Chapter 4 already discussed the equipments in use and the control strategies implemented in pumping systems while working at off-design conditions. To control the flow in a centrifugal pump, the head can be reduced through valve throttling or adjusting the frequency of the electric motor until the operating conditions allow pump to deliver the desired flow rate. As there is no direct control over the flow rate, the operating point can be changed only by means of changing the head or the rotational speed of the pump. In terms of numerical simulations, the use of velocity-inlet and pressure-outlet boundary conditions become unrealistic, as the flow rate is controlled directly. On the other hand, a setup with pressure-inlet and pressure-outlet conditions is more suitable to study the pump behaviour under more realistic circumstances.

In this study, the behaviour of the wastewater pump due to different control strategies is assessed through unsteady numerical simulations with the use of dynamic boundary conditions by means of user-defined functions. A detailed description of the codes implementation is found in Appendix A.

8.1 Pressure control

In order to assess the energy performance with throttling control, the numerical model with the modified geometry was run for twenty revolutions at eight prescribed static pressure outputs ranging from 50 kPa to 160 kPa in 15 kPa increments. The rotational speed was set at the nominal design speed of 1475 rpm. The performance metrics of such operating points were already reported in Fig. 5.12 and compared with experimental data. In the current study, the unsteady solution is given as an input and a target static pressure (target head) is set to be achieved within five different time scales. The simulations are further run for twenty additional revolutions to let the system going back to periodic behaviour. During the transient phase, “Head velocity”, or head variation rate coefficient $\dot{H}$, is defined as:

$$\dot{H} = \frac{H_f - H_i}{\Delta t_i}$$  \hspace{1cm} (8.1)
Chapter 8. Transient performance with variable speed drives at off-design conditions

Where $H_f$ is the target head, $H_i$ is the input head and $\Delta t_i$ is the flow time, in seconds, of the transient period. The transient period is the amount of impeller rotations needed to achieve the target head, from five to twenty-five with intervals of five revolutions. The head variation rate $\dot{H}$ has units $\text{m} \cdot \text{s}^{-1}$, and indicates how quickly the system achieves a target head. Figure 8.1a shows the variation, according to different head variation rates, from the initial to the target head. As expected, the higher head variation rate is associated with the steepest curve at $\dot{H}=15 \text{ m} \cdot \text{s}^{-1}$. This data is directly gathered from the prescribed pressure-outlet boundary condition, which varies linearly with the time step.

![Figure 8.1](image_url)

**Figure 8.1:** Variation of pump performance metrics at different head variation rates: a) Head with flow time, b) Instantaneous flow rate with flow time, c) Flow rate with flow time averaged over each impeller revolution ($\text{△} - \dot{H}=3 \text{ m} \cdot \text{s}^{-1}$; $\text{-} - \text{□} - \dot{H}=5 \text{ m} \cdot \text{s}^{-1}$; $\text{-} - \text{●} - \dot{H}=7.4 \text{ m} \cdot \text{s}^{-1}$; $\text{-} - \text{○} - \dot{H}=15 \text{ m} \cdot \text{s}^{-1}$; $\text{-} - \text{△} - \dot{H}=3 \text{ m} \cdot \text{s}^{-1}$).
After the transient, the steady state solution shows that the head is constant over time. Conversely, Figure 8.1b shows the instantaneous flow rate and Figure 8.1c shows the flow rate averaged each impeller revolution. As expected, as the head increased the flow rate decreases. The oscillating behaviour for the flow rate seen in Fig. 8.1b is directly connected to the unsteady pressure fluctuations within the pump vane due to the rotation of the impeller, as seen in Fig. 5.7. In order to have a more meaningful comparison between head and flow rate in the transient behaviour, the non-dimensionalised head $H^*$ and the non-dimensional flow rate $Q^*$ are reported with the non-dimensional flow time $t^*$ in Fig. 8.2.

![Figure 8.2](image)

**Figure 8.2:** Non-dimensional pump performance with head variation rates: a) Non-dimensional head with non-dimensional flow time, b) Non-dimensional flow rate with non-dimensional flow time ($\cdots$ $H=15\text{ m}\cdot\text{s}^{-1}$; $\nabla$ $H=7.4\text{ m}\cdot\text{s}^{-1}$; $\rightarrow$ $H=5\text{ m}\cdot\text{s}^{-1}$; $\leftarrow$ $H=3.7\text{ m}\cdot\text{s}^{-1}$; $\triangle$ $H=3\text{ m}\cdot\text{s}^{-1}$).

The quantities are non-dimensional with respect to the average value at the end of the transient period, as follows:

\[
H^* = \frac{H(t)}{H_t} \tag{8.2}
\]

\[
Q^* = \frac{Q(t)}{Q_t} \tag{8.3}
\]

\[
t^* = \frac{t}{t_t} \tag{8.4}
\]
Where $H_t$, $Q_t$ and $t_t$ are reference quantities with respect to the head, the flow rate and the flow time values at the end of the transient. The use of such nomenclature was preferred over flow and head coefficients as it refers to a transient period, whereas flow and head coefficients usually address steady operating conditions. It is observed that the non-dimensional head curves clearly show a linear behaviour and overlap on top of each other for both the duration of the transient and steady state periods. Figure 8.2b shows the variation of the non-dimensional flow rate with the non-dimensional flow time according to the head variations seen in Fig. 8.1a. Comparing Fig. 8.2a and Fig. 8.2b, it is clear that head and flow rate exhibit different transient behaviours. While the non-dimensional head curves are overlapped, the flow rate pattern becomes less linear as the head variation rate increases. The faster the system is forced to change its operating point, the longer it takes to achieve periodic behaviour at the steady state. This is observed by looking at the blue curve ($H=15 \text{ m}^{-1}$) which is above all the others for $t < 1.3 \text{ s}$ and below for $1.3 \text{ s} < t < 2 \text{ s}$ before achieving periodic behaviour.

While the input of the system (head) is specified to vary linearly in time, the response (flow rate) is clearly non-linear. This behaviour is perhaps explained by considering that the faster the system is forced to change its operating point, the greater the inertia of the mass flow and therefore, the system will be slower to achieve periodic behaviour. As a result, the overall power of the pump is increased with the head variation rate.

**Figure 8.3:** Variation of the power with flow time at different head variation rates: a) Hydraulic power, b) Shaft power time ($\Box H=15 \text{ m}^{-1}$; $\blacktriangledown H=7.4 \text{ m}^{-1}$; $\lozenge H=5 \text{ m}^{-1}$; $\circ H=3.7 \text{ m}^{-1}$; $\triangle H=3 \text{ m}^{-1}$).
8.1. Pressure control

Figure 8.3 shows the hydraulic and the shaft power with the flow time during the transient for each head variation rate, averaged for each impeller revolution. It is observed that both powers increase at the beginning of the transient and decrease afterwards. It is clear that the greater is the head variation rate, the higher the power spike during the first impeller revolutions. This behaviour is perhaps explained by the need to draw more power in order to increase the pump head to achieve the new operating point in a shorter number of rotations. On the other hand, as the head variation rate increases, the lower the power at the end of the transient. This is perhaps due to the increased inertia of the flow rate at higher head variation rates, as discussed earlier for the non-dimensional quantities.

![Figure 8.3](image)

Figure 8.4: Variation of performance metrics at different head variation rates:
(a) Shaft power, (b) Mechanical energy.

Figure 8.4 compares the shaft power and the total energy during the transient with the head variation rate. Figure 8.4a reports the maximum value of the mechanical power during the transient. It is observed that the power absorbed by the pump increases linearly with the head variation rate, from 9.31 kW at $\dot{H}=3 \text{ m} \cdot \text{s}^{-1}$ to 9.36 kW at $\dot{H}=15 \text{ m} \cdot \text{s}^{-1}$. On the other hand, the mechanical energy $E_m$, shown in Fig. 8.4b shows a totally different behaviour. The mechanical energy is defined as the integral of the mechanical power during the transient as follows:

$$E_m = \int_{t_0}^{t_f} P_m \, dt \quad (8.5)$$
Chapter 8. Transient performance with variable speed drives at off-design conditions

Where \( t_0 \) and \( t_t \) are the flow time at the beginning and at the end of the transient, respectively. It is clear from Fig. 8.4b that as the head variation rate increases, the mechanical energy decreases exponentially. This is mainly due to the amount of time required by the system to achieve the new operating point. It is concluded that when the operating point of a centrifugal pump is accomplished by only valve throttling, a faster transition will require the pump to spend more power to increase the head, but less energy during the transient. Conversely, a slower transition will require less power but the energy consumed during the transient will be higher.

8.2 Combined pressure-speed control

To assess the behaviour of the wastewater pump while using variable speed drives, the computational model was run for different rotational speeds ranging from 80% to 120% of the nominal design speed \( \omega_n \), with 10% intervals. The pressure-outlet boundary condition was initially varied from 55 kPa to 220 kPa with 15 kPa intervals.

<table>
<thead>
<tr>
<th>Rotational speed [rpm]</th>
<th>Relative rotational speed</th>
<th>Time step size [s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \omega_1 = 1180 )</td>
<td>80% ( \omega_n )</td>
<td>( \Delta t = 4.23 \times 10^{-4} )</td>
</tr>
<tr>
<td>( \omega_2 = 1327.5 )</td>
<td>90% ( \omega_n )</td>
<td>( \Delta t = 3.76 \times 10^{-4} )</td>
</tr>
<tr>
<td>( \omega_3 = \omega_n = 1475 )</td>
<td>100% ( \omega_n )</td>
<td>( \Delta t = 3.39 \times 10^{-4} )</td>
</tr>
<tr>
<td>( \omega_4 = 1622.5 )</td>
<td>110% ( \omega_n )</td>
<td>( \Delta t = 3.08 \times 10^{-4} )</td>
</tr>
<tr>
<td>( \omega_5 = 1770 )</td>
<td>120% ( \omega_n )</td>
<td>( \Delta t = 2.82 \times 10^{-4} )</td>
</tr>
</tbody>
</table>

Table 8.2: System curves design parameters.

<table>
<thead>
<tr>
<th>System curve</th>
<th>Design flow rate [m³·h⁻¹]</th>
<th>Reynolds number</th>
<th>friction factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>( h_A )</td>
<td>200</td>
<td>( 4.74 \times 10^5 )</td>
<td>0.0145</td>
</tr>
<tr>
<td>( h_B )</td>
<td>100</td>
<td>( 2.37 \times 10^5 )</td>
<td>0.016</td>
</tr>
</tbody>
</table>

Because of the different rotational speed of each data set with respect to the design speed \( \omega_n \), the time step size was varied according to Eq. 4.20 and reported in Table 8.1. Two different system curves are added to the model to simulate the presence of a looping one-dimensional line attached to the wastewater pump. System curve \( h_A \) is designed to allow the pump to run at the maximum efficiency at different rotational speeds, while a system curve \( h_B \) is designed to allow the pump to run at lower efficiency values. To facilitate the design of the system curves, a unique value of the flow rate is assumed to calculate the pipeline Reynolds numbers and smooth pipes made of stainless steel (relative roughness \( 1 \times 10^{-4} \)) for the calculation of the friction factors. Table 8.2 shows the design parameters of the designed system curves. The performance metrics (e.g. head-discharge, torque and efficiency) with the system curves are compared with experimental data, as shown in
8.2. Combined pressure-speed control

Fig. 8.5. The experimental data refers to the design rotational speed of 1475 rpm. The values from the numerical simulations are averaged over the final revolutions. Figure 8.5a shows the head-discharge curve for the rotational speed ranging from $\omega_1$ to $\omega_5$.

![Graph](image)

**Figure 8.5:** Variation of pump performance metrics with volumetric flow rate for different rotational speeds: a) Pressure Head-Flow rate, b) Average torque about impeller rotation axis, c) Hydraulic efficiency (□ Experimental data; ◊ $\omega_1 = 80\% \omega_D$; ▲ $\omega_2 = 90\% \omega_D$; □ $\omega_3 = 100\% \omega_D$; □ $\omega_4 = 110\% \omega_D$; △ $\omega_5 = 120\% \omega_D$; –– system A curve $h_A$ —— system B curve $h_B$.)
The curve trends show consistency with previous numerical outcomes and reported experimental data. As expected, a reduction in the rotational speed yields to lower flow rates for the same pressure-outlet values. As a result, the head-discharge curves are shifted to the left, with comparable slopes. The head-discharge curve at $\omega_4 = 110\% \omega_D$ shows the least discrepancy with experimental data. Therefore, the discrepancy between numerical simulations and experimental data at design speed is approximately 10% for all examined operating conditions. The lower number of data points shown at lower rotational speed is caused by the fact that as the pressure increased, the numerical simulations calculated negative values of the flow rates. Due to the unrealistic nature of this result, it is more likely that the pump model was incurring in flow reversal do to the impossibility of operating at shut-off. Therefore, it is concluded that when the rotational speed of the pump is decreased, the shut-off condition is achieved for lower values of the outlet pressure. The shut-off head is the head a pump develops when operating against a closed discharge valve. This condition occurs when it is necessary to purge all the air within the piping system in order to reduce the pressure fluctuations within the piping. Figure 8.5b and c shows the torque and the hydraulic efficiency, respectively. As for previous numerical simulations, the trend for the numerical torque shows good agreement with experimental data, being the minimum at $\omega_1 = 80\% \omega_D$ and the maximum at $\omega_5=120\% \omega_D$.

The hydraulic efficiency in Fig. 8.5c is specified as the ratio between the difference of the hydraulic power between inlet and outlet to the shaft power at the impeller. The effect of increased rotational speed is to shift the maximum efficiency towards higher flow rates. It is observed that the efficiency is 71.8% at $\omega_1 = 80\% \omega_D$ to 68% at $\omega_5 = 120\% \omega_D$. These values are over-predicted by 0.1% at $\omega_1$ and under-predicted by 3.5% at $\omega_1$ with respect to the experimental data at design speed. Figure 8.5c also shows the system curve A intersecting the above-mentioned points from $147 \text{m}^3\cdot \text{h}^{-1}$ to $256 \text{m}^3\cdot \text{h}^{-1}$. The system curve B intersects the points at $61 \text{m}^3\cdot \text{h}^{-1}$ and $131 \text{m}^3\cdot \text{h}^{-1}$, where the computed efficiency is 51% at $\omega_1$ and 55.2% at $\omega_5$, respectively. The operating points intersected by the system curves are analysed in more detail to assess the pump energy performance when the rotational speed is varied from $\omega_1$ to $\omega_5$. The torque, rotational speed and efficiency values according to the operating conditions identified by the two system curves are shown in Fig. 8.6 for three head variation rates ($\dot{H}=31.15 \text{m} \cdot \text{s}^{-1}$, $\dot{H}=11.7 \text{m} \cdot \text{s}^{-1}$, $\dot{H}=7 \text{m} \cdot \text{s}^{-1}$). In order to assess the transient behaviour in the operating points constrained through the one-dimensional pipeline, the outlet pressure is varied from 55 kPa to 120 kPa for the off-design operating points along $h_A$ and between 78 kPa to 175 kPa for the operating points along $h_B$. The rotational speed ranges from 1180 rpm to 1770 rpm. The dynamic boundary conditions in the UDF codes are assumed to vary linearly as function of the time step (i.e. flow time). The inlet pressure and the rotational speed increments each time step are calculated to achieve the final steady values in 5, 15 and 25 revolutions, corresponding to three different head variation rate values: $\dot{H} = 35.45 \text{m} \cdot \text{s}^{-1}$, $\dot{H} = 11.7 \text{m} \cdot \text{s}^{-1}$ and $\dot{H} = 7 \text{m} \cdot \text{s}^{-1}$, respectively. Figure 8.6a shows the variation of the rotational speed with the flow rate for different head variation rates. The operating conditions are relative to the path followed by the continuous and the dotted lines in Fig. 8.5a and Fig. 8.5c representing the system curve B and the system curve A, respectively.
8.2. Combined pressure-speed control

Figure 8.6: Variation of performance metrics at different head variation rates ($h_A$, right; $h_B$, left): a) Rotational speed, b) Torque, c) Efficiency ($\square$ $H = 35.45 \text{ m/s}^1$; $\Diamond$ $H = 11.7 \text{ m/s}^1$; $\circ$ $H = 7 \text{ m/s}^1$).
As expected, higher values of the rotational speed are observed during the transient with increased head variation rate. Moreover, while the rotational speed is a linear function of the flow time, the flow rate behaviour in the transient is clearly non-linear. This phenomenon is emphasized as much as the pump operates far from the best efficiency conditions, which is clearly seen in the trend for \( h_B \). At the onset of the transient, the flow rate increases from 144 m\(^3\)·h\(^{-1}\) to 423 m\(^3\)·h\(^{-1}\) along \( h_A \) and from 82 m\(^3\)·h\(^{-1}\) to 77 m\(^3\)·h\(^{-1}\) along \( h_B \), and increases afterwards for both systems. It should be remembered that while the rotational speed increases, increasing the volumetric flow rate, the static pressure also increases, which typically yields to lower flow rates. One possible explanation of this phenomenon is here provided. At the onset, the increase in the static pressure causes a reduction in the flow rate. Meanwhile, the impeller acceleration must overcome the fluid inertia, which is increased as the flow is decelerating because of the rise in head. It could be that for the first rotations, the effect of changing the pressure head is prevalent with respect to the change in the rotational speed. This effect is emphasized at higher head variation rates because of the strongest pressure gradients. As the operating conditions keep changing during the transient, the effect of increasing the rotational speed becomes more effective on the flow rate than the static pressure raise, thus volumetric flow rate increases. It is also observed that the efficiency is reduced from 71.8% to 68.7% along the system A curve and from 68.5% to 53.5% along the system B curve. This reduction is the result of reduced hydraulic power added to the system with respect to the same mechanical power input. While the torque and the rotational speed both increase with the flow time, the flow rate decreases, and the hydraulic power is reduced. The efficiency loss is limited when the operating point is changed within a higher number of revolutions (i.e. slower), but the drop is more substantial when the pump is running at off-design. It is also observed that along \( h_A \) the efficiency is almost constant and decreases with increased head variation rate.

The behaviour of hydraulic efficiency with the flow rate is shown in Fig. 8.6c for different head variation rates. It is observed that the change in the operating point results in a lower efficiency as the head variation rate increases. Along the system curve \( h_B \), the efficiency drop is quantified to be 4.5% at \( \dot{H} = 7 \text{ m·s}^{-1} \) and 13% at \( \dot{H} = 35.45 \text{ m·s}^{-1} \) with respect to the periodic value. Along the system A curve, the efficiency up to 11.2% at \( \dot{H} = 7 \text{ m·s}^{-1} \) and 19.1% at \( \dot{H} = 35.45 \text{ m·s}^{-1} \). The efficiency loss is due to the lower value of the hydraulic power during the transient with respect to the mechanical power. This is caused by the fact that while the torque and the rotational speed both increase with the flow time, the flow rate is seen to decrease during the transient, yielding even lower values of the hydraulic efficiency. A change in the operating point by means of speed and valve control shows a loss in efficiency both at design and off-design conditions. However, the loss in efficiency is limited when the transition is conducted in a higher number of revolution. Figure 8.7 reports the maximum averaged mechanical power and the mechanical energy for the two operating conditions during the transient. It is observed that the power decreases by only 0.04 kW from 10.95 kW and 0.02 kW from 10.57 kW along \( h_1 \) and \( h_2 \), respectively. Moreover, the discrepancy in the power consumption between the two different operating conditions is found to be 0.4 kW. The mechanical power is calculated as the torque by the angular velocity, which is constant for both the operating conditions. On the other hand, the duration of the transient increases by five
8.3. Transient behaviour with the gap size

times, meaning more energy consumed at lower head variation rates. A similar behaviour is observed when the pump is running off-design conditions.

\[
\begin{align*}
\text{\textbf{Figure 8.7:} Variation of performance metrics at different head variation rates:} \\
\text{a) Shaft power, b) Mechanical energy (□ h}_1; \diamond h}_2). \\
\end{align*}
\]

It is concluded that a change in the operating point by means of both speed and pressure control to increase both rotational speed and the head is more efficient when accomplished through a higher number of impeller revolutions, but it is found to be more energy intensive. On the other hand, achieving the new operating point in fewer impeller revolutions will consume less mechanical energy during the transient, but will result in a less efficient transition.

8.3 Transient behaviour with the gap size

The analysis reported in Chapter 2 showed a reduction in the energy performance due to different impeller-wear plate axial distances. In the context of assessing the transient behaviour of the wastewater pump for off-design conditions, the computational models corresponding to gap sizes of \( s_g = 0.3 \, \text{mm}, 0.6 \, \text{mm}, 1.15 \, \text{mm} \) and \( 2 \, \text{mm} \) were run for the same set of rotational speed and static pressure outlet conditions as for section 8.2. The resulting performance metrics in terms of pressure head, torque and efficiency curves with flow rate at different gap size and rotational speed, here not reported, show similar behaviour as for the characteristic curves at \( s_g = 0.3 \, \text{mm} \) in Fig. 8.5.
Figure 8.8: Variation of performance metrics at $H = 35.45 \, \text{m} \cdot \text{s}^{-1}$ (h$_A$, right; h$_B$, left): a) Head, b) Torque, c) Efficiency ($\circ s_g = 0.3 \, \text{mm}; \Box s_g = 0.6 \, \text{mm}; \ast s_g = 1.15 \, \text{mm}; \bigtriangleup s_g = 2 \, \text{mm}$).
The observed drop in performance due to increased gap size is consistent with the results found in Chapter 6. The outcomes for the transient simulations with the gap size for a head variation rate of $\dot{H} = 35.45 \text{ m}\cdot\text{s}^{-1}$ are shown in Fig. 8.8. As for Fig. 8.7, the right-hand side curves are computed along the system curve $h_A$, while the left-hand side curves are computed along $h_B$. Figure 8.8a shows the variation of the head with the flow rate along the two system curves. The right-hand side curves represent the operating points along the system A curve $h_A$, and it is relative to lower values of the pressure head. On the other hand, the left-hand side curves are relative to the system B curve $h_B$, thus higher value of the head are expected. It is observed that for a similar set of operating conditions, the flow rate and the efficiency are reduced as the gap size is increased. It is worth mentioning that while in Fig. 7.1b the torque decreases with the axial clearance for constant flow rate, in Fig. 8.8 the torque is insensitive to the gap size, and the developed flow rate is reduced for increased gap due to leakage around the impeller tip. The two case analysed show different behaviour: along the system curve A, the flow rate always increases during the transient for constant $s_g$, while along system curve B, the flow rate initially decreases, increasing afterwards. It is worth to note that for the system curve B, as the gap size increases, the flow rate developed at the end of the transient is approximately the same as at the onset. For instance, at $s_g = 2 \text{ mm}$, the flow rate slightly increases from $50.1 \text{ m}^3\cdot\text{h}^{-1}$ to $50.89 \text{ m}^3\cdot\text{h}^{-1}$, despite the additional mechanical energy spent because of an increment in the torque by almost three times. The drop in the flow rate during the transient could be due to the same reasons discussed before for the transient behaviour in section 8.2. However, the flow rate is further reduced because of the leakage mechanism with the gap size. As a result, the hydraulic efficiency, shown in Fig. 8.8, is reduced. It is observed that the reduction follows a downward trend, shifting towards lower values of efficiency and developed flow rate. This behaviour is experienced for all the examined cases, and it is due to the lower value of the hydraulic power, due to the drop in the flow rate, with respect to the mechanical power input at the impeller shaft, which increases with the transient flow time.

8.4 Scaling parameters

The following analysis examines the performance characteristics of the pump in terms of non-dimensional coefficients to assess the transient behaviour for nine different operating conditions when either the rotational speed, the pressure head, or both, are changed with different head variation rates. To maintain similarity in the head, flow and power coefficients, three different pressure outlet values ($p_1 = 70 \text{ kPa}$, $p_2 = 77.8 \text{ kPa}$, $p_3 = 63 \text{ kPa}$) and three rotational speeds ($\omega_1 = 1475 \text{ rpm}$, $\omega_2 = 1554.8 \text{ rpm}$, $\omega_3 = 1639 \text{ rpm}$) were used as boundary conditions. Figure 8.9 shows the non-dimensional coefficients at constant head variation rate $\dot{H} = 3.5 \text{ m}\cdot\text{s}^{-1}$. The three different operating conditions involve the head coefficient to range between to constant values, from $C_H = 0.1306$ to $C_H = 0.1176$, as shown in Fig. 8.9a. While the flow coefficient in Fig. 8.9a shows similar behaviour for all the examined cases, Fig. 8.9b shows a drop in the flow coefficient when the operating point is changed by means of speed control and combined pressure-speed control. This flow coefficient (i.e. flow rate) reduction could be caused by the same phenomena as for the flow rate reduction in
Fig. 53b, due to the flow inertia. In fact, as shown in Fig. 56b, the flow rate reduction is higher when the static pressure is increased. It is also observed that the flow coefficient does not achieve steady state immediately at the end of the transient, as the flow coefficient. This is the same effect that was shown in Fig. 49b for the non-dimensional flow rate and it may be caused by the non-linear response of the flow rate. Figure 8.9c and Fig. 8.9d show the power coefficient and the efficiency, respectively. When the rotational speed is varied, the power coefficient and the efficiency present an oscillating behaviour with increased power consumption and reduced efficiency when speed and combined pressure-speed controls are employed. As the data is monitored at the outlet pipe, it is an indicator of the correlation between the pipeline and the pump dynamic due to the incompressible flow.

![Figure 8.9: Non-dimensional coefficients for $H = 3.5 \text{ m.s}^{-1}$: a) Head coefficient $C_H$, b) Flow coefficient $C_Q$, c) Power coefficient $C_W$, d) Hydraulic efficiency $\eta$ (○ $\omega_1$ to $\omega_2$, $p_1$ = const; □ $\omega_1$ to $\omega_2$, $p_1$ = const; △ $\omega_1$ to $\omega_3$, $p_1$ to $p_3$).](image)

The efficiency drops from 65% to 61% when the operating conditions are varied by means of changing only the rotational speed with respect to solely increasing the outlet head (i.e. valve closing), but it is further reduced to 60% when both rotational speed and head are varied.
8.5 Summary

This chapter discussed the transient performance at off-design conditions when different control strategies are employed to vary the operating conditions of the wastewater pump. Static pressure, rotational speed and gap size were varied to address the energy performance with different time scales to assess whether a faster or slower change in the operating point was suitable in terms of energy consumption. The computational model was coupled with a one-dimensional pipeline by intersecting the performance curves with two different system curves, and the adoption of different control strategies was discussed. The more relevant findings can be resumed as following:

• When the operating conditions are varied by means of increasing only the pressure at the inlet (i.e. valve control), the flow rate showed non-linearities involving a higher settling time to achieve periodic behaviour. A head variation rate coefficient was defined to quantify the rate of change in head with the number of revolutions during the transient. It was found that the mechanical peak power increased with the head variation rate, meaning that a higher amount of electricity is drawn when the operating point is changed faster, and less energy is required when the transition is slower;

• The performance curves of the wastewater pump were obtained for a range of rotational speed from 80% to 120% of the nominal value. The transient conditions along two system curves with combined pressure-speed control were investigated, finding that the efficiency drops with the head variation rate. The cause of inefficiencies were addressed to the reduction of the flow rate due to the increased inertia of the fluid, as the increase in the static pressure counteracts the increase in the rotational speed at the beginning phase of the transient;

• The mechanical power for both systems was found to be insensitive to the head variation rate, while the energy decreases with increased head variation rate. Since the efficiency shows dependency with the system curves and head variation rates, a faster transition will be more efficient and energy intensive, while a slower transition will consume less energy during the transient but it will be less efficient;

• The transient simulations at off-design conditions showed that the energy performance decreases with increased gap size. The reduction in the energy performance for pressure-speed control is found to be due to the combination of increased tip leakage due to the gap with the inertial effects due to transient boundary conditions;

• Non-dimensional flow, head and power coefficients were defined to study the transient characteristics between two operating point for multiple combination of pressure and rotational speed. It was found that when the pump is controlled by means of changing the rotational speed, the dynamics is influenced by an oscillating behaviour of the head. This is due to the non-linear response of the flow rate causing increased energy consumption and reduced efficiency during the transient.
Chapter 9

Conclusions

This thesis assesses the energy performance of a wastewater pump for increasing impeller-casing gap size at transient off-design operating conditions. A three-dimensional computational model of the PE-2 XFP 150E CB1.1 was developed using ANSYS Workbench from a manufactured design of the pump. The modified geometry was used to achieve a compromise between numerical accuracy and computational effort. Fluent 19.2 was used as a numerical solver.

Unsteady simulations using a modified geometry were performed at different operational regimes using velocity-inlet and pressure-outlet boundary conditions. A contour plot of the pressure field at different impeller positions shows good agreement with the general behaviour of centrifugal pumps found in literature. Stagnation points at the blade trailing edge and at the volute tongue were predicted. The displacement of the stagnation points with increased flow rate was found to be in agreement with published data. Pressure fluctuations at the outlet were found to vary with an oscillating trend according to the trailing edge position of the rotating impeller within the vane. The peak amplitude was found to decrease with increased flow rate because of the stronger turbulent intensity at higher flow rates. The performance metrics in terms of head-discharge, torque and efficiency curves were found to be consistent with experimental data. The numerical simulations relative to the modified geometry were found to be more reliable with respect to the original geometry, providing a more stable solution for a higher range of operating conditions. The results obtained using pressure-inlet, pressure-outlet boundary conditions were found to show consistency with experimental data and good agreement with previous numerical assessments.

The orbit Fig. 6.1 did not show oscillations because of the one-way coupling method of the numerical model, implying no deflections of the impeller structure. The behaviour of the non-dimensional radial forces with the gap size shown that tip leakages cause a reduction of the force amplitude along the x-axis while the radial force along the z-axis is insensitive to increasing gap size. The eccentricity of the orbit curves is found to decrease with increasing the rotational speed of the impeller. This is due to the lower pressure fluctuations caused by the leakage mechanism. A cross-correlation between the non-dimensional forces was calculated to calculate a time lag as phase shift, finding that the force $F_z$ is delayed by $84^\circ$ with respect to $F_x$. A poly-harmonic function applied to the force time series found that amplitude of the force $F_y$ at the fundamental harmonics is four times smaller than $F_x$ and $F_z$. On the other hand, the amplitude of the non-dimensional force
Chapter 9. Conclusions

$F_y$ is higher at for all subsequent harmonics. These results were found to be consistent with published data, offering further confidence in the numerical model.

The modified geometry was varied to assess the energy performance of the wastewater pump with increasing impeller-wear plate gap size. The pump performance in terms of head-discharge, torque and hydraulic efficiency curves were run for five gap sizes at six flow rates for a total of 30 different operating points. It was found that the energy performance drop with increasing the gap size. This is manifested as a drop of the back pressure at the inlet interface, caused by leakage around the tip of the impeller, with the largest drop in efficiency reported as 13%. Non-dimensional sensitivity factors were defined as the partial derivative of the assessed quantity with respect to its reference value to quantify the drop in performance per unit of axial gap. The non-dimensional coefficients $K$ and $K'$ suggest that efficiency and mechanical power can drop up to 13.5%·mm$^{-1}$ and 0.58 kW·mm$^{-1}$, respectively. An assessment of the velocity components through the gap allowed for further understanding of the mechanism of leakage. While the axial and the radial velocity represents the swirling motion of the fluid particle from the pressure side to the suction side, the absolute velocity magnitude is mainly driven by the rotational speed. The pressure distribution in the gap region indicates that at lower heads leakage become more relevant and therefore, back-flow is increased.

The computational model using the modified geometry was run with time-varying boundary conditions to assess the variation of the pump performance with coupled and segregated pressure-speed control. A head rate coefficient was determined to calculate how fast the operating point is changed. The mechanical power is found to increase with the head rate only when a pressure control is applied. When the new operating point is achieved using a lower number of impeller revolutions, the total mechanical energy input is lower. On the other hand, a higher input power is observed. The performance curves of the wastewater pump were obtained for a rotational speed ranging from 80% to 120% of the nominal design speed. The intersection of the head-discharge curves with system curves allowed to define the operating points in one-dimensional pipeline systems. The transient characteristics relative to two system curves were investigated, finding that non-linearities in the flow rate behaviour were responsible for inefficiencies. The mechanical power was found to be insensitive to the head rate, while the energy input decreased with the head rate. The simulations relative to combined pressure-speed control with increased gap size found the energy performance to be reduced with increasing the gap size. Non-dimensional flow, head and power coefficients were used to investigate the transient behaviour when a new operating point is achieved at a different combination of pressure and rotational speed values. It was found that when the rotational speed is varying, the power coefficient and the efficiency oscillate. This is due to the non-linear response of the flow rate, which behaves as a second-order system increasing the energy consumption and reducing the hydraulic efficiency during the transient.

The high fidelity computational model developed in this work enabled the assessment of the performance of a wastewater pump at off-design conditions with good agreement with respect to experimental data and previous numerical studies. It is also found that the transient behaviour does not have a substantial impact on the overall performance of the pump. However, it is found
that the transition strategy between two operating conditions has a significant impact on the peak power and the total energy input. The study of the axial gap between the impeller and the wear-plate showed that the mechanism of leakage causes a significant drop in pump performance. For this reason, it is essential to include specific maintenance protocols to ensure axial clearance is maintained within design targets, as specified by pump manufacturers.

### 9.1 Future developments

While the above mentioned results will help filling some of the gaps discovered in the existing literature, some other investigations are here proposed to explore further areas of current research:

- The performance data of the wastewater pump in chapter 5 showed that discrepancies in the head-discharge, torque and efficiency curves are mainly due to the assumption of isotropic turbulence model, which is typical of \( k-\varepsilon \) models. It could be of interest to minimize the discrepancy between experimental and numerical data by using turbulence models based on anisotropic assumption of the turbulent flow, such as Reynolds Stress Transport (RST) models;

- The study of the effects of the gap size on the energy performance showed an increase in the efficiency for a particular set of operating conditions. This was addressed as potential physical phenomenon that could give rise to non-synchronous vibrations. However, the time and spatial scale of current settings were not small enough to capture such behaviour, that could be investigated with the use of SST \( k-\omega \) or LES turbulence models. However, based on the current \( y^+ \) value and Courant number, this would require an increase in the CPU time by a factor of 6 and an increase in the mesh size by a factor of 5 to achieve \( y^+ = 1 \). This would also help for a better understanding of the mechanism of leakage through the gap;
Appendix A

UDF codes

The codes here reported were used to solve the transient simulations with time-variant boundary conditions. This is accomplished by specifying a user-defined function (UDF), which is then loaded into the numerical solver. The following functions enable the variation of the static pressure and the rotational speed of the impeller and are written using C++ programming language.

A.1 Static pressure UDF

This user-defined function allows the user to dynamically vary the static pressure at the inlet interface by loading it as a pressure-outlet boundary condition. The case file was first run to achieve periodic behaviour at constant pressure (e.g. initial pressure). Then, the initial pressure was increased by a step pressure (delta P) each time step up to the chosen final pressure. The initial pressure was increased by a step deltaP equal to the ratio of difference between the final and the initial pressure by the amount of time steps during the transient. In the code below, the equation for the outlet static pressure is a function of the flow time and varies linearly with the time step. The variable ts is the actual time step, while t0 is the time step at the beginning of the simulation and tt is the number of time steps at the end of the transient. A loop was required to specify a maximum number of time steps where the pressure was increased. This was required to allow the computational model to keep a constant pressure outlet downstream the transient simulation.

```c
#include "udf.h"

DEFINE_PROFILE(unsteady_pressure, thread, position)
{
  face_t f;
  real ts = N_TIME;
  begin_f_loop(f, thread)
  {
    if (ts<tt)
      F_PROFILE(f, thread, position) = Initial pressure + delta_P*(ts-t0);
    else
```
Appendix A. UDF codes

A.2 Rotational speed UDF

This user-defined function allows the user to dynamically vary the rotational speed of the impeller and the inlet pipe zones by loading it into the cell-zones boundary conditions. Similar to the UDF for the outlet pressure, the equation for the rotational speed is a function of the time and varies linearly with the time step from an initial speed at constant value. The initial speed was increased by a step delta\_e equal to the ratio of the difference between the final and the initial speed by the amount of time steps during the transient. In the code below, the variables t0, ti and tt have the same meaning as for the pressure outlet UDF. As the rotational speed was specified in rpm at all times, the variable “rate” was used to allow the conversion from rpm to rad/s.

```c
#include "udf.h"
DEFINE_ZONE_MOTION(fmotion,omega,axis,origin,velocity,time,dtime)
{
real ts = N_TIME;
real rate= 0.10472;
if (ts<tt)
{
*omega = (initial speed + delta_N*(ts-t0))*rate;
}
else
{
*omega = final speed*rate;
}
N3V_D(velocity,=,0.0,0.0,0.0);
N3V_S(origin,=,0.0);
N3V_D(axis,=,0.0,-1.0,0.0);
return;
}
```
Appendix B

Computational methods

Numerical modelling involves the resolution of a set of algebraic or differential equations to be solved with computer-aided numerical techniques. Numerical simulations are required to study the behaviour of those systems for which the analytical solution is too complex, as in most common non-linear systems. Computational approach is a relatively new discipline to solve physical problems in engineering design and analysis of natural phenomena. Of those methodologies concerning the study of fluids, CFD is certainly the most widely employed in industry and research institutions. Of those existing techniques to address the fluid mechanics of centrifugal pumps, Finite Volume Method (FVM) is widely employed in CFD commercial packages.

A mathematical model representative of a physical phenomenon as fluid flow is based on the resolution of the flux balance equation:

\[ \frac{\partial u}{\partial t} + \nabla \cdot \Gamma = F \text{ in } \Omega \]  \hspace{1cm} (B.1)

Where \( u \) denotes a physical quantity such as momentum, mass or energy, \( \Gamma \) denotes the flux of that quantity, \( F \) is a source and \( \Omega \) is the domain. Weighting the equations with test functions \( \varphi \), and integrating over the model domain yields:

\[ \int_{\Omega} \frac{\partial u}{\partial t} \varphi \, dV + \int_{\Omega} (\nabla \cdot \Gamma) \varphi \, dV = \int_{\Omega} F \varphi \, dV \]  \hspace{1cm} (B.2)

Applying the divergence theorem to \( \Gamma \varphi \) and skipping intermediate passages, the weak formulation of the finite element methods can be written as:

\[ \int_{\Omega} \frac{\partial u}{\partial t} \varphi \, dV - \int_{\Omega} \Gamma \cdot \nabla \varphi \, dV + \int_{d\Omega} \Gamma \cdot n \, dS = \int_{\Omega} F \varphi \, dV \]  \hspace{1cm} (B.3)

Where the third term on the left-hand side integrates the flux of \( u \) and \( \Gamma \) over the domain boundary \( d\Omega \). A special case is that one for a constant test function \( \varphi = 1 \), for which eq. B.3 is:

\[ \int_{\Omega} \frac{\partial u}{\partial t} \, dV + \int_{d\Omega} \Gamma \cdot n \, dS = \int_{\Omega} F \, dV \]  \hspace{1cm} (B.4)
This relation is used for finite volume methods and constitute a particular case of the more generic weak formulation for finite elements method, in eq. B.3. The main difference is about the discretization: eq. B.3 is obtained from picking a finite number of test functions \( \varphi = \varphi_h \), while eq. B.4 requires picking a finite number of control volumes \( \Omega = \Omega_h \).

In most finite elements methods, test functions are only non-zero in the vicinity of so-called nodes (or vertices), thus integrals are computed only in the elements adjacent to the node, shown in Fig B.1a. For internal nodes, as that surrounded by grey triangles, all the adjacent elements give contribution to the domain integral over \( \Omega \). For nodes at the boundary, the two adjacent blue give flux contribution to the integral over \( \partial \Omega \) through the third term in eq. B.3, while the blue and the light green give contribution to the domain integral. In the finite volume method, each cell is treated as an individual domain. The second term in eq B.4 is integrated for all cells, both internal and external, that have a face (3D) or an edge (2D) on the boundary.

(a) Finite elements method integration. (b) Finite volume method integration.

Figure B.1: FEM and FVM approaches [Fontes, 2018].

The physical variable \( u \) and the flux \( \Gamma \) have different approximations between the two approaches. In the finite elements method, the same basic function is used to approximate the solution as test function and as long as the solution has a polynomial degree higher than zero, the first-order derivatives can be approximated. In the finite volume methods the solution at the boundary is not well defined and needs to interpolate the solution and the flux over a higher number of cells. However, two drawbacks of the finite elements method can be mentioned: first, only global conservation is guaranteed, i.e. only the net flux over the domain boundaries is guaranteed to be in balance. Second, there is no control on local fluxes, thus for some problems (e.g. convection-dominated flow) the weak formulation must be modified, yielding to more expensive computational costs. Both methods possess high accuracy and only measuring the CPU time and memory requirements to solve a fluid problems is the correct way to select the best approach to be used for a particular problem. Traditionally, numerical models involving fluids have always been studied with finite volume methods, although some commercial packages (e.g. COMSOL) are working to improve FEM-base approaches (e.g. discontinuous Galerkin) for more efficient computing.

The numerical works presented in the literature showed a variety of software capable of studying
centrifugal pumps, although the majority make use of the ANSYS suite. This software allows to study a high range of phenomena, from single to multi-phase flows, isothermal or reacting, compressible or not, including the widely used and well-validated ANSYS CFX and ANSYS Fluent which have been extensively used to test centrifugal pump performance. Both software are FVM- and user-friendly GUI-based, making use of a grid for spatial discretization and possess high reliability in terms of accuracy, each having its own advantages compared to the other. The most significant differences between the two is in the way they integrate the fluid flow equations and the solving strategies: ANSYS CFX solver is vertex-centered while ANSYS Fluent is cell-centered: this is the location of the unknown to be solved within the fluid domain. In a cell-centered method, the cell serves as control volume to store the average variable value while in the vertex-centered method, the control volume is formed combining sub-control volumes surrounding the vertex, as shown in Fig. B.1.

Comparisons can be done by measuring CPU, memory, accuracy and grid flexibility requirements. Cell-centered method has a larger number of degrees of freedom but less fluxes per unknown, yielding to higher computational cost when compared to the vertex-centered method, but similar accuracy. On the other hand, Fluent offers more options from a solver point of view (both coupled and segregated), while CFX is only a coupled solver. Despite that, they are very similar in terms of software capabilities. Acharya [Acharya, 2016] carried out a study comparing Fluent and CFX for a test case to assess the software differences in terms of convergence characteristics. Bulk quantities (e.g. pressure, velocity) showed discrepancies in magnitude between the two software when the same turbulence model was used, which could be due the different discretization method and wall functions. Nevertheless, there is no evidence suggesting which solver provided the most reliable results. On the other hand, CFX showed better convergence in terms of computational time and stability. The same was observed by Stenmark [Stenmark, 2013] who assessed multiphase flow using CFX and Fluent on a vertical T-Junction. Moreover, multiphase modelling is often used to simulate cavitation in centrifugal pumps. Despite the slower convergence, Fluent showed improved accuracy regarding the volume fraction and gas velocity profiles, thanks to the higher customization embedded in the solver.

For turbomachinery, the majority of researchers seem to be oriented towards CFX for its simplicity.
Appendix B. Computational methods

and friendly GUI, while Fluent is preferred for compressible flow or more general applications. The numerical simulations in the current work were carried using Fluent v.19.2 thanks to the superior customization options and higher user expertise with such software.

B.1 Governing equations

The physics principles for the motion of viscous fluid substances are based on the conservation of mass, Newton’s second law, i.e. the conservation of momentum, and the conservation of energy. This collection of equations composes the Navier-Stokes equations, which are a set of coupled, hyperbolic-parabolic partial differential equations, which is the most general description of fluid mechanics. As real flow is inherently three-dimensional, involves turbulence, and depends strongly on the initial and boundary conditions to the respective problem, the equations can only be solved analytically for very few, highly simplified flows. Generally, a numerical procedure is required to approximate the solution and involves a vast consumption of computational resources. As the smallest scales in space and time are determined by the turbulent fluctuations of the flow, the predominant reduction in effort can be made assuming turbulent fluctuations. Any flow variable, representing a scalar or vector components, is therefore split to a mean and a fluctuating part:

$$ \Phi = \bar{\Phi} + \Phi' $$

by definition, the average of the fluctuating part is zero:

$$ \bar{\Phi}' = \lim_{t \to \infty} \frac{1}{T_0} \int_0^T \Phi' dt = 0 $$

Replacing the latter equation yields the Reynolds-Averaged Navier-Stokes equations (RANS). In fluids, the conservation laws apply within a certain spatial region known as the control volume (CV), where its extensive properties such as mass, momentum and energy, relates the rate of changes of the amount of that property to externally determined effects. Considering the flow variable \( \phi \) as the intensive property (\( \phi = 1 \) for mass conservation, \( \phi = v \) for momentum conservation and \( \phi = e \) for conservation of energy (\( e \) is the internal energy), the corresponding extensive property \( \Phi \) can be written as:

$$ \Phi = \int_{\Omega} \rho \phi \, d\Omega $$

Where \( \Omega_{CV} \) is the control volume. Using this definition, any left side of the conservation law is a substantial derivative and can be written as:

$$ \frac{D\Phi}{Dt} = \frac{d}{dt} \int_{\Omega_{CV}} \rho \phi \, d\Omega = \frac{d}{dt} \int_{\Omega_{CV}} \rho \phi \, d\Omega + \int_{S_{CV}} \rho \phi (\bar{v} - \bar{v}_b) \cdot \hat{n} \, dS $$

where \( S_{CV} \) is the surface closing CV, \( \hat{n} \) is the unit vector orthogonal to each dS, and directed outwards, \( \bar{v} \) is the fluid velocity vector and \( \bar{v}_b \) is the velocity at which dS is moving. This equation states
that the rate of change of an extensive property for the system of fluid particles that occupy the control volume at a given instant of time, is the sum of the rate of change of that property within the control volume plus the flux of it through the volume boundaries due to the fluid motion relative to the boundary. This last term is better known as *convective* flux of $\phi$ through CV boundary.

**B.1.1 Conservation of mass**

The conservation of mass states that matter may neither be created nor destroyed, so it must be conserved within the control volume:

$$\frac{\partial}{\partial t} \int_{\omega_{CV}} \rho \, d\Omega + \int_{s_{CV}} \rho \vec{v} \cdot \vec{n} \, dS = 0 \quad (B.9)$$

The above equation states that the time variation of the mass within the control volume is equal to the mass flux through the control surface. In terms of substantial derivative, the general form of the conservation of mass is:

$$\frac{\partial \rho}{\partial t} + (\vec{v} \cdot \nabla) \rho + \rho \nabla \cdot \vec{v} = 0 \quad \rightarrow \quad \frac{D\rho}{Dt} + \rho \nabla \cdot \vec{v} = 0 \quad (B.10)$$

However, for incompressible flows the rate change of the density $\rho$ is zero, so $D\rho/Dt=0$. Therefore, only the last term of Eq B.10 remains, identifying the continuity equation for incompressible flows (solenoidal field):

$$\rho \nabla \cdot \vec{v} = 0 \quad (B.11)$$

**B.1.2 Conservation of momentum**

The derivation of Newton’s second law applied to an infinitesimal element of a continuous fluid states that the time variation of the momentum is equal to the sum of all the body forces $\vec{f}_b$ plus the sum of all the surface forces $\vec{f}_s$ applied to to the system of fluid particles that occupy the CV at a certain instant:

$$\frac{\partial}{\partial t} \int_{\omega_{CV}} \rho \vec{v} \, d\Omega + \int_{s_{CV}} \rho \vec{v} (\vec{v} \cdot \vec{n}) \, dS = \sum \vec{f}_b + \sum \vec{f}_s \quad (B.12)$$

The Eq. B.12 can also be written as:

$$\frac{\partial}{\partial t} \int_{\omega_{CV}} \rho \vec{v} \, d\Omega + \int_{s_{CV}} \rho \vec{v} (\vec{v} \cdot \vec{n}) \, dS = \int_{\omega_{CV}} \rho \vec{f} \, d\Omega + \int_{s_{CV}} \vec{t} \cdot \vec{n} \, dS \quad (B.13)$$

Where $\vec{T}$ the stress tensor, which is the molecular rate transport of momentum, that for Newtonian fluids can be written as:

$$\vec{T} = -p \cdot \vec{I} + \vec{\sigma} = - \left( p \cdot \vec{I} + \frac{2}{3} \mu \nabla \cdot \vec{v} \right) + 2 \mu \vec{D} \quad (B.14)$$
Appendix B. Computational methods

Where \( \mu \) is the dynamic viscosity, \( \mathbf{I} \) is the unit tensor, \( p \) is the static pressure and \( \mathbf{D} \) is the rate of strain (deformation) tensor. For incompressible flows, the fluid viscosity is assumed to be isotropic. Consequently, only the last part of Eq. B.14 survives, which is called Stokes’ stress constitutive equation
\[
[\mathbf{D}] = 2\mu \mathbf{V}
\]
where \( \mathbf{V} \) is the rate of strain tensor. For incompressible flows, the fluid viscosity is assumed to be isotropic. Consequently, only the last part of Eq. B.14 survives, which is called Stokes’ stress constitutive equation
\[
\tau = 2\mu \mathbf{D}
\]

Rewriting Eq. B.13 in terms of substantial derivative as Eq. B.10:

\[
\rho \frac{D \mathbf{V}}{Dt} = \frac{\partial p}{\partial t} + \mathbf{V} \cdot \nabla p + \mu \nabla^2 \mathbf{V} - \sum \mathbf{f}_b
\]

Where \( \sum \mathbf{f}_b \) and \( \rho \mathbf{g} \) are additional body forces, i.e. by a coupled disperse phase, and the gravitational force respectively. The former can be assumed negligible having a single-phase fluid. The Reynolds-averaging leaves the expression of \( \mathbf{F} \) in Eq. B.14 unclosed and the remaining part of the stress tensor, that depends on the mean turbulent fluctuations, is therefore called the Reynolds-stress tensor and needs to be modelled by semi-empirical closure terms. The momentum equation for the i-th dimension under explicit declaration of the Reynolds-stresses \( -\rho \mathbf{u}_i' \mathbf{u}_j' \) is given by:

\[
\rho \frac{D \mathbf{V}_i}{Dt} = \rho \mathbf{g} \cdot \mathbf{u} + \mathbf{v} \cdot \nabla (\rho \mathbf{u}) + \mu \nabla^2 \mathbf{V}
\]

where \( \delta_{ij} \) is the Kronecker-delta (\( \delta_{ij} = 1 \) for \( i=j \) and \( \delta_{ij} = 0 \) otherwise). Eq. B.17 can be expressed as substantial derivative:

\[
\rho \frac{D \mathbf{V}}{Dt} = \rho \mathbf{g} - \nabla p + (\mu + \lambda) \mathbf{V} \cdot \nabla \mathbf{V} + \mu \nabla^2 \mathbf{V}
\]

Where if \( \lambda \) and \( \mu \) are constant, the fluid is defined as Newtonian. For incompressible flows, where the divergence of \( \mathbf{V} \) is zero \( \nabla \cdot \mathbf{V} = 0 \), replacing the conservation of mass equation, the latter becomes:

\[
\rho \frac{D \mathbf{V}}{Dt} = \rho \mathbf{g} - \nabla p + \mu \nabla^2 \mathbf{V}
\]

Which is the simplest form for the conservation of momentum.

B.1.3 Conservation of energy

For compressible flows, the total energy of a control volume is the sum of the internal energy increase by the heat added to the system minus the energy increase due to the work done by the fluid element, which satisfies the second principle of thermodynamics:

\[
\frac{D e}{Dt} = \rho \mathbf{g} \cdot \mathbf{V} + \mathbf{V} \cdot \left( \mathbf{V} \right) + \rho \mathbf{q} - \nabla \cdot \mathbf{k}
\]

Where \( \mathbf{q} \) is the heat per unit mass and \( \mathbf{k} \) is the heat loss by conduction. For incompressible flows, the energy conservation law is decoupled from the other conservation laws, as the only influence is implied in the temperature-dependent material properties. For water, the temperature rise does
not depend on the pressure rise but only from the viscous forces between the fluid particles and the walls, whose contribution is negligible.
Appendix C

Publications

C.1 Journal Articles

Appendix D

Datasheet PE90/4-50Hz XFP150ECB1
Submersible Sewage Pump
Type ABS XFP 80C - 201G

Robust, reliable, submersible pumps, with Premium Efficiency motors from 1.3 to 25.0 kW. For the pumping of wastewater and sewage from buildings and sites in private, commercial, industrial and municipal areas.

Features
- The water-pressure-tight, encapsulated, flood-proof motor and the pump section form a compact, robust, modular construction.
- NEMA Class A temperature rise.
- Premium Efficiency motors in accordance with IEC 60034-30 level IE3 with testing in accordance with IEC60034-2-1.
- Continuously rated motor in submerged and non-submerged applications.
- Double mechanical seals; SiC-SiC at the medium side, SiC-C at the motor. All seals are independent of rotation direction and resistant to temperature shock.
- Anti-wicking cable plug solution (80C - 150E), or water-pressure-sealed connection chamber (100G - 201G).
- Hydraulic options of Contrablock and Contrablock Plus impellers for high efficiency, or vortex impellers for maximum solids handling.
- Lubricated-for-life bearings with a calculated lifetime of minimum 50,000 hrs. (80C - 150E), and 100,000 hrs. (100G - 201G).
- Stainless steel shaft. Designed with high safety factor to prevent fatigue fracture.
- Temperature monitoring by thermal sensors (140 °C) in the stator windings.
- Seal monitoring by a moisture probe (DI) in the seal chamber (80C - 150E), or dry chamber (100G - 201G), which signals an inspection alert if there is leakage at the shaft seals.
- Smooth outer design to reduce rag build-up.
- Stainless steel lifting hoop.
- DN 80, DN 100, DN 150 and DN 200 radial slot DIN flange discharge.
- Maximum allowable temperature of the medium for continuous operation is 40 °C.
- Maximum submergence depth of 20 m.
- Explosion-proof as standard, in accordance with international standard ATEX II 2G Ex db IIB T4 Gb.

Motor
Premium Efficiency IE3, three-phase, squirrel-cage motor; 400 V, 50 Hz; 2-pole (2900 r/min), 4-pole (1450) and 6-pole (980). Protection type IP 68, with stator insulation Class H.
Start-up: 1.3 - 3.0 kW = direct on line (DOL)
4.0 - 25.0 kW and 3.0 kW 6-pole = star-delta (YΔ).
Service factor: 1.3
Motors with other operating voltages and frequencies are also available.

Identification Code: e.g. XFP 80C CB1.3 PE22/4-C-50
Hydraulics:
XFP .......... Product range
8 ............ Discharge outlet DN (cm)
0 .......... Hydraulic type
C .......... Volute opening (dia. mm)
CB .......... Impeller type: CB = Contrablock, VX = vortex
1 .......... Number of impeller vanes
3 .......... Impeller size
Motor:
PE .......... Premium Efficiency
22 .......... Motor power P2 kW x 10
4 .......... Number of poles
C .......... Volute opening (dia. mm)
50 .......... Frequency
## Technical data

### XFP

<table>
<thead>
<tr>
<th>XFP</th>
<th>Motor</th>
<th>Impeller size</th>
<th>Rated voltage</th>
<th>Motor power* ((\text{W}))</th>
<th>Motor current ((\text{A}))</th>
<th>Speed ((\text{r/min}))</th>
<th>Cable size</th>
<th>Weight** ((\text{kg}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>PE 200/2</td>
<td>3, 4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 200/2</td>
<td>1, 2</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 100/2</td>
<td>4, 5, 6, 7</td>
<td>400–230</td>
<td>1.8</td>
<td>1.5</td>
<td>3.2</td>
<td>1450</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 200/2</td>
<td>3, 4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 29/4</td>
<td>1</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 100/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 100/2</td>
<td>1, 2</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 40/2</td>
<td>1</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>2</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 110/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 110/2</td>
<td>1, 2</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 40/2</td>
<td>1</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 110/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 110/2</td>
<td>1, 2</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 40/2</td>
<td>1</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 110/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 110/2</td>
<td>1, 2</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 40/2</td>
<td>1</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 110/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 110/2</td>
<td>1, 2</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 40/2</td>
<td>1</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 110/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 110/2</td>
<td>1, 2</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 40/2</td>
<td>1</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
<tr>
<td>PE 30/2</td>
<td>4</td>
<td>400–230</td>
<td>2.3</td>
<td>2.2</td>
<td>3.4</td>
<td>12.1</td>
<td>100 G1.5</td>
<td>100 / n.a.</td>
</tr>
</tbody>
</table>

* \(P_1\) = power at mains; \(P_2\) = power at motor shaft. **Without / with cooling jacket; includes 10 m cable. Data for alternative voltages available on request.

Appendix D. Datasheet PE90/4-50Hz XFP150ECB1

137
### Standard and options

<table>
<thead>
<tr>
<th>Description</th>
<th>Standard</th>
<th>Option</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mains voltage</td>
<td>400 V 3~</td>
<td>230, 500, 695 V *</td>
</tr>
<tr>
<td>Voltage tolerance</td>
<td>± 10%</td>
<td></td>
</tr>
<tr>
<td>Motor efficiency</td>
<td>Premium Eff. IE3</td>
<td>-</td>
</tr>
<tr>
<td>Insulation class</td>
<td>H</td>
<td>-</td>
</tr>
<tr>
<td>Start-up</td>
<td>Direct on line (DOL), star-delta (YΔ)</td>
<td>-</td>
</tr>
<tr>
<td>Approvals</td>
<td>Ex / ATEX</td>
<td>-</td>
</tr>
<tr>
<td>Mechanical seal (at medium side)</td>
<td>SIC-SIC-NBR</td>
<td>SIC-SIC-Viton</td>
</tr>
<tr>
<td>Mechanical seal (at motor side)</td>
<td>SIC-C-NBR</td>
<td></td>
</tr>
<tr>
<td>O-rings</td>
<td>NBR</td>
<td>VITON (external seals)</td>
</tr>
<tr>
<td>Cables</td>
<td>HO7RN8-F</td>
<td>EMC</td>
</tr>
<tr>
<td>Cable length (m)</td>
<td>10</td>
<td>20, 30, 40, 50</td>
</tr>
<tr>
<td>Protective coating</td>
<td>2k Epoxy 120 μm</td>
<td>2k Epoxy 400 μm</td>
</tr>
<tr>
<td>Preparation for lifting hoist</td>
<td>Lifting hoop</td>
<td>-</td>
</tr>
<tr>
<td>Cooling</td>
<td>Self-cooling (80C - 150E); by the medium (100G - 201G)</td>
<td>Closed cooling (100G - 201G)</td>
</tr>
<tr>
<td>Installation</td>
<td>Wet-wall</td>
<td>Dry wall or transportable</td>
</tr>
</tbody>
</table>

* Selected motors only. Contact Sulzer for details.

### Monitoring

<table>
<thead>
<tr>
<th>Description</th>
<th>Standard</th>
<th>Option</th>
</tr>
</thead>
<tbody>
<tr>
<td>Motor (temperature)</td>
<td>Bi-metallic switch in windings</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>PTC thermistor in windings</td>
<td>-</td>
</tr>
<tr>
<td>Seals (leakage)</td>
<td>Moisture sensor (DI) in oil chamber (80C - 150E)</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Moisture sensor (DI) in dry chamber (100G - 201G)</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>Moisture sensor (DI) in connection chamber (100G - 201G)</td>
<td>-</td>
</tr>
</tbody>
</table>

Temperature and leakage relays are required. See accessories table.

### Materials

<table>
<thead>
<tr>
<th>Description</th>
<th>Material</th>
<th>Option</th>
</tr>
</thead>
<tbody>
<tr>
<td>Motor housing</td>
<td>Cast iron EN-GJL-250</td>
<td>-</td>
</tr>
<tr>
<td>Volute</td>
<td>Cast iron EN-GJL-250</td>
<td>Ceramic coated EN-GJL-250**</td>
</tr>
<tr>
<td>Impeller</td>
<td>Cast iron EN-GJL-250</td>
<td>Stainless steel 1.4470 (AISI 329)<strong>, Flame hardened or ceramic coated EN-GJL-250</strong></td>
</tr>
<tr>
<td>Bottom plate</td>
<td>Cast iron EN-GJL-250</td>
<td>Stainless steel 1.4470 (AISI 329)<strong>, Flame hardened or ceramic coated EN-GJL-250</strong></td>
</tr>
<tr>
<td>Motor shaft</td>
<td>Stainless steel 1.4021 (AISI 430)</td>
<td>-</td>
</tr>
<tr>
<td>Lifting hoop</td>
<td>Stainless steel 1.4401 (AISI 316)</td>
<td>-</td>
</tr>
<tr>
<td>Fasteners</td>
<td>Stainless steel 1.4401 (AISI 316)</td>
<td>-</td>
</tr>
</tbody>
</table>

** Selected models only. Contact Sulzer for details.
### Accessories

<table>
<thead>
<tr>
<th>Description</th>
<th>Size</th>
<th>XFP</th>
<th>Part no.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Fixed installation - wet well with Sulzer Automatic Coupling System</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pedestal* (cast iron EN-GJL-250)</td>
<td>90º cast bend (single guide rail) - DIN flange connection</td>
<td>DN 80</td>
<td>80C - 81E</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 100</td>
<td>100C - 100G</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 100 (high-head)</td>
<td>101G</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 150</td>
<td>150E - 150G</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 200</td>
<td>200G</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 200</td>
<td>201G</td>
</tr>
<tr>
<td>Pedestal* (cast iron EN-GJL-250)</td>
<td>90º cast bend (single guide rail) - plug/clamp connection</td>
<td>DN 100 pipe Ø60 mm</td>
<td>80C - 81E</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 100 pipe Ø109 mm</td>
<td>100C - 100G</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 100 high head Ø110G mm</td>
<td>101G</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 150 pipe Ø160 mm</td>
<td>150E - 150G</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 200</td>
<td></td>
</tr>
<tr>
<td>Pedestal* (cast iron EN-GJL-250)</td>
<td>90º cast bend (twin guide rail) - DIN flange connection</td>
<td>DN 80</td>
<td>80C - 81E</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 100</td>
<td>100C - 101G</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 150</td>
<td>150E - 150G</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DN 200</td>
<td>200G &amp; 201G</td>
</tr>
<tr>
<td>Pedestal bracket fasteners</td>
<td>single guide rail version (galvanised steel)</td>
<td>80C - 81E</td>
<td>62160325</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100C - 101G</td>
<td>62160326</td>
</tr>
<tr>
<td></td>
<td></td>
<td>150E - 150G</td>
<td>62160327</td>
</tr>
<tr>
<td></td>
<td></td>
<td>200G &amp; 201G</td>
<td>62160328</td>
</tr>
<tr>
<td></td>
<td>single guide rail version (stainless steel)</td>
<td>80C - 81E</td>
<td>62160329</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100C - 101G</td>
<td>62160330</td>
</tr>
<tr>
<td></td>
<td></td>
<td>150E - 150G</td>
<td>62160331</td>
</tr>
<tr>
<td></td>
<td></td>
<td>200G &amp; 201G</td>
<td>62160332</td>
</tr>
<tr>
<td></td>
<td>twin guide rail version (galvanised steel)</td>
<td>80C - 81E</td>
<td>62160333</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100C - 101G</td>
<td>62160334</td>
</tr>
<tr>
<td></td>
<td></td>
<td>150E - 150G</td>
<td>62160335</td>
</tr>
<tr>
<td></td>
<td></td>
<td>200G &amp; 201G</td>
<td>62160336</td>
</tr>
<tr>
<td></td>
<td>Pedestal base anchor bolts</td>
<td>single and twin guide rail (galvanised steel)</td>
<td>80C - 101E</td>
</tr>
<tr>
<td></td>
<td></td>
<td>150E - 150G</td>
<td>62160776</td>
</tr>
<tr>
<td></td>
<td></td>
<td>200G &amp; 201G</td>
<td>62160777</td>
</tr>
<tr>
<td></td>
<td>Chain Kit (galvanised steel)</td>
<td>including shackle</td>
<td>3 m</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4 m</td>
<td>61265070</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6 m</td>
<td>61265071</td>
</tr>
<tr>
<td></td>
<td></td>
<td>7 m</td>
<td>61265072</td>
</tr>
<tr>
<td></td>
<td>Chain Kit (stainless steel)</td>
<td>including shackle</td>
<td>3 m</td>
</tr>
<tr>
<td></td>
<td></td>
<td>4 m</td>
<td>61265096</td>
</tr>
<tr>
<td></td>
<td></td>
<td>6 m</td>
<td>61265102</td>
</tr>
<tr>
<td></td>
<td></td>
<td>7 m</td>
<td>61265103</td>
</tr>
<tr>
<td><strong>Fixed installation - dry well, (horizontal)</strong>*</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pump Support Kit (EN-GJL-250)</td>
<td>head and volute supports with fixing bolts and vibration damper</td>
<td>80C, 81C.</td>
<td>61825023</td>
</tr>
<tr>
<td></td>
<td></td>
<td>80C, 81C, 100C.</td>
<td>61825022</td>
</tr>
<tr>
<td></td>
<td></td>
<td>80E.</td>
<td>61825020</td>
</tr>
<tr>
<td></td>
<td></td>
<td>81E.</td>
<td>61825021</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100C.</td>
<td>61825025</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100E.</td>
<td>61825026</td>
</tr>
<tr>
<td></td>
<td></td>
<td>150E, 151E.</td>
<td>61825031</td>
</tr>
<tr>
<td></td>
<td></td>
<td>101G.</td>
<td>61825032</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100G, 101G, 150G, 200G, 201G.</td>
<td>61825033</td>
</tr>
<tr>
<td>Ground Support Stand</td>
<td></td>
<td>80C.</td>
<td>61355014</td>
</tr>
<tr>
<td></td>
<td></td>
<td>80E &amp; 81E.</td>
<td>61355020</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100C.</td>
<td>61355015</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100E.</td>
<td>61355019</td>
</tr>
<tr>
<td></td>
<td></td>
<td>150E, 151E.</td>
<td>61355021</td>
</tr>
<tr>
<td></td>
<td></td>
<td>101G.</td>
<td>61355022</td>
</tr>
<tr>
<td>Adapter kit</td>
<td>required with support stand</td>
<td>80C.</td>
<td>62685347</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100C.</td>
<td>62685349</td>
</tr>
<tr>
<td><strong>Transportable</strong></td>
<td>Skirtbase</td>
<td>80C, 81C, 100C.</td>
<td>61355016</td>
</tr>
<tr>
<td></td>
<td></td>
<td>80E &amp; 81E.</td>
<td>61355017</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100C.</td>
<td>61355018</td>
</tr>
<tr>
<td></td>
<td></td>
<td>100E.</td>
<td>61355020</td>
</tr>
<tr>
<td></td>
<td></td>
<td>150E, 151E.</td>
<td>61355019</td>
</tr>
<tr>
<td></td>
<td></td>
<td>101G.</td>
<td>61355022</td>
</tr>
<tr>
<td><strong>General</strong></td>
<td>Cathodic Protection (zinc anodes)</td>
<td>80C - 201G.</td>
<td>13905000</td>
</tr>
<tr>
<td>Leakage Relay Type ABS CA 461</td>
<td>110 - 230VAC</td>
<td>80C - 201G</td>
<td>16907010</td>
</tr>
<tr>
<td></td>
<td>18 - 36VDC, SELV</td>
<td>16907011</td>
<td></td>
</tr>
<tr>
<td>Temperature and Leakage Relay Type ABS CA 462</td>
<td>110 - 230VAC</td>
<td>80C - 201G</td>
<td>16907006</td>
</tr>
<tr>
<td></td>
<td>18 - 36VDC, SELV</td>
<td>16907007</td>
<td></td>
</tr>
</tbody>
</table>

*Guide rail not included **Vortex version of pumps (VX) *** Contrablock version of pump (CB)
XFP150E CB1 50HZ

Operating data specification

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow</td>
<td></td>
</tr>
<tr>
<td>Efficiency</td>
<td></td>
</tr>
<tr>
<td>NPSH</td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td>20 °C</td>
</tr>
<tr>
<td>No. of pumps</td>
<td>1</td>
</tr>
</tbody>
</table>

Pump data

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>XFP150E CB1 50HZ</td>
</tr>
<tr>
<td>Series</td>
<td>XFP PE1-PE3</td>
</tr>
<tr>
<td>No. of vanes</td>
<td>1</td>
</tr>
<tr>
<td>Free passage</td>
<td>100 mm</td>
</tr>
<tr>
<td>Discharge flange</td>
<td>DN150</td>
</tr>
<tr>
<td>Moment of inertia</td>
<td>0.0444 kg m²</td>
</tr>
</tbody>
</table>

Motor data

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rated voltage</td>
<td>400 V</td>
</tr>
<tr>
<td>Rated power P2</td>
<td>9 kW</td>
</tr>
<tr>
<td>Number of poles</td>
<td>4</td>
</tr>
<tr>
<td>Power factor</td>
<td>0.79</td>
</tr>
<tr>
<td>Starting current</td>
<td>118 A</td>
</tr>
<tr>
<td>Starting torque</td>
<td>121 Nm</td>
</tr>
<tr>
<td>Insulation class</td>
<td>H</td>
</tr>
<tr>
<td>Frequency</td>
<td>50 Hz</td>
</tr>
<tr>
<td>Nominal Speed</td>
<td>1470 1/min</td>
</tr>
<tr>
<td>Efficiency</td>
<td>90.8 %</td>
</tr>
<tr>
<td>Rated current</td>
<td>18.1 A</td>
</tr>
<tr>
<td>Rated torque</td>
<td>58.6 Nm</td>
</tr>
<tr>
<td>Degree of protection</td>
<td>IP 68</td>
</tr>
<tr>
<td>No. starts per hour</td>
<td>15</td>
</tr>
</tbody>
</table>

Sulzer reserves the right to change any data and dimensions without prior notice and can not be held responsible for the use of information contained in this software.
Motor performance curve

Motor PE90/4-E-50HZ

Rated power: 9 kW
Nominal Speed: 1470 1/min
Number of poles: 4
Rated voltage: 400 V
Date: 2017-12-14

Frequency: 50 Hz

<table>
<thead>
<tr>
<th>Symbol</th>
<th>No load</th>
<th>25 %</th>
<th>50 %</th>
<th>75 %</th>
<th>100 %</th>
<th>125 %</th>
</tr>
</thead>
<tbody>
<tr>
<td>P₂ / kW</td>
<td>0</td>
<td>2,25</td>
<td>4,5</td>
<td>6,75</td>
<td>9</td>
<td>11,25</td>
</tr>
<tr>
<td>P₁ / kW</td>
<td>0,4971</td>
<td>2,677</td>
<td>4,949</td>
<td>7,373</td>
<td>9,911</td>
<td>12,54</td>
</tr>
<tr>
<td>η / %</td>
<td>0</td>
<td>84,05</td>
<td>90,92</td>
<td>91,55</td>
<td>90,81</td>
<td>89,72</td>
</tr>
<tr>
<td>n / 1/min</td>
<td>1500</td>
<td>1495</td>
<td>1487</td>
<td>1477</td>
<td>1466</td>
<td>1453</td>
</tr>
<tr>
<td>cos φ</td>
<td>0,07976</td>
<td>0,3756</td>
<td>0,5875</td>
<td>0,7203</td>
<td>0,7912</td>
<td>0,8295</td>
</tr>
<tr>
<td>I / A</td>
<td>8,996</td>
<td>10,29</td>
<td>12,16</td>
<td>14,77</td>
<td>18,08</td>
<td>21,82</td>
</tr>
<tr>
<td>s / %</td>
<td>0</td>
<td>0,3251</td>
<td>0,863</td>
<td>1,521</td>
<td>2,265</td>
<td>3,113</td>
</tr>
<tr>
<td>M / Nm</td>
<td>0</td>
<td>14,37</td>
<td>28,9</td>
<td>43,64</td>
<td>58,62</td>
<td>73,92</td>
</tr>
</tbody>
</table>

Tolerance according to VDE 0530 T1 12.84 for rated power

Starting current: 118 A
Starting torque: 121 Nm
Moment of inertia: 0,0449 kg m²
No. starts per hour: 15

Sulzer reserves the right to change any data and dimensions without prior notice and can not be held responsible for the use of information contained in this software.
Bibliography


[IRENA, 2019] IRENA (2019). Climate change and renewable energy: National policies and the role of communities, cities and regions (report to the g20 climate sustainability working group (cswg)).


