Aerodynamic characteristics of circular cylinder and squared cylinders with and without rounded corners

Hugo Rafael Lopes Abreu

Thesis to obtain the Master of Science Degree in

Aerospace Engineering

Supervisors: Prof. Luís Rego da Cunha de Eça
Dr. Guilherme Nuno Vasconcelos Beleza Vaz

Examination Committee
Chairperson: Prof. Filipe Szolnoky Ramos Pinto Cunha
Supervisor: Prof. Luís Rego da Cunha de Eça
Member of the Committee: Prof. Pedro António Martins Mendes

December 2015
Resumo

Escoamentos viscosos em torno de cilindros é um tema de estudo clássico na dinâmica de fluidos computacional (CFD), com uma grande quantidade de aplicações práticas no domínio da aerodinâmica e da hidrodinâmica. Muitas aplicações de engenharia usam cilindros em que as seções transversais variam de circulares a quadradas com cantos arredondados. Números de Reynolds típicos de aplicações práticas estão no intervalo de \(10^5\) a \(10^6\) onde ocorre a chamada “drag crisis”. As simulações de escoamentos nestas condições são extremamente desafiantes, porque o escoamento apresenta regiões laminares, de transição e turbulentas. Além disso, devido à existência de libertação de vórtices, o escoamento não é estatisticamente permanente.

Embora as equações de Navier-Stokes em média de Reynolds (RANS) complementadas com um modelo de viscosidade turbulenta tenham limitações evidentes para modelar escoamentos tão complexos, existem vários estudos publicados na literatura para simular este tipo de escoamentos com este modelo matemático. Devido à natureza periódica da libertação dos vórtices, médias de conjunto devem ser utilizadas para a definição do escoamento médio e para o cálculo da média do balancê de massa e da quantidade de movimento, o que é normalmente designado por URANS.

Esta tese apresenta um estudo do escoamento em torno de um prisma quadrangular com cantos arredondados baseados em soluções numéricas das equações URANS para escoamento bidimensional e incompressível. Selecionamos o modelo de viscosidade turbulenta \(k – \omega\) SST amplamente utilizado em aplicações práticas de engenharia. Dois exercícios distintos são apresentados: a investigação da influência do tamanho do domínio de cálculo e a especificação das condições de fronteira para a pressão e para as variáveis dependentes do modelo de turbulência; a simulação do escoamento a diferentes números de Reynolds para identificar os regimes do escoamento. Todos os cálculos são efectuados com o solver ReFRESCO e estudos de refinamento, malha e passo temporal, e de convergência iterativa são realizados para todas as situações analisadas para estimar a incerteza numérica.

Os resultados obtidos mostram uma influência significativa das dimensões do domínio e das condições de fronteira aplicadas à pressão e/ou às variáveis dependentes do modelo de turbulência. Os erros de discretização para o nível de refinamento utilizado é significativo e o erro iterativo em cada passo no tempo teve de ser reduzido significativamente abaixo das habituais três ordens de grandeza de redução dos resíduos. Apesar do erro numérico não ser desprezável, o simples modelo matemático adoptado é capaz de identificar a influência do número de Reynolds nas propriedades do escoamento.

**Palavras-chave:** 2D prisma quadrangular, cantos arredondados, URANS, libertação de vórtices, tamanho do domínio computacional, condições de fronteira, efeito do número de Reynolds
Abstract

Viscous flows around cylinders are a classical research topic in computational fluid dynamics (CFD) with a vast amount of practical applications in the field of aerodynamic and hydrodynamic. Many engineering applications use cylinders that range from circular cross-sections to square cylinders with rounded corners. Typical Reynolds numbers of practical applications are in the range of $10^5$ to $10^6$ where the so-called “drag crisis” occurs. Flow simulations in such conditions are extremely challenging because the flow exhibits laminar, transitional and turbulent regions. Furthermore, due to the existence of vortex shedding, the flow is not statistically steady.

Although the Reynolds-Averaged Navier-Stokes (RANS) equations supplemented by eddy-viscosity models have evident shortcomings in such complex flows, there are several attempts published in the open literature to simulate this type of flows with such mathematical model. Due to the periodic nature of vortex shedding, ensemble averaging must be used for the definition of the mean flow and for the averaging of the mass and momentum balance. Therefore, the RANS equations are not statistically steady, which is usually designated by URANS.

This thesis presents a study of the flow around a square cylinder with rounded corners based on the numerical solution of the URANS equations for bidimensional incompressible flow. We have selected the shear stress transport (SST) $k-\omega$ eddy-viscosity model which is widely in practical engineering applications. Two different exercises are presented: an investigation of the influence of the size of the computational domain and of the pressure and turbulence quantities boundary conditions; the simulation of the flow at different Reynolds numbers to identify the flow regimes. All calculations are performed with the solver ReFRESCO. Grid/time refinement and iterative convergence studies are performed for all flow conditions to estimate the numerical uncertainty.

The results obtained show a significant dependence on the size of the computational domain and on the pressure and turbulence quantities boundary conditions. For the present level of grid/time refinement there is a significant discretization error and the iterative convergence criteria used at each time step must be a lot more demanding than the usual three orders of magnitude of residual drop. Although the numerical uncertainty is not negligible, this simple mathematical model is able to capture the influence of the Reynolds number on the different flow regimes.

Keywords: 2D square cylinder, rounded corners, URANS, vortex shedding, computational domain size, boundary conditions, Reynolds number effect
# Table of Contents

Resumo ................................................................. iii
Abstract ............................................................... v
List of Tables ......................................................... x
List of Figures ......................................................... xiv
Nomenclature ......................................................... xvii

1 Introduction ......................................................... 1
  1.1 Motivation ....................................................... 1
  1.2 Flow around cylinders ......................................... 2
    1.2.1 Forces acting on a cylinder ................................ 2
    1.2.2 Vortex shedding ........................................... 3
  1.3 Objectives ...................................................... 6
  1.4 Structure of the thesis ......................................... 8

2 Mathematical Model ............................................... 9
  2.1 Flow regime .................................................... 9
    2.1.1 Reynolds-Averaged Navier-Stokes equations ................. 9
    2.1.2 Turbulence model ......................................... 11
  2.2 Domain size .................................................... 12
  2.3 Boundary conditions ........................................... 13
  2.4 Quantities of interest ......................................... 14

3 Numerical Solution ................................................ 15
  3.1 ReFRESCO flow solver .......................................... 15
  3.2 Test case ....................................................... 15
    3.2.1 Geometry .................................................. 15
    3.2.2 Grid generation ........................................... 16
  3.3 Grid sets and time step ........................................ 17
  3.4 Calculation details ............................................ 19

4 Sensitivity studies ................................................ 21
  4.1 Numerical uncertainty .......................................... 21
4.1.1 Iterative errors ......................................................... 21
4.1.2 Discretization errors ................................................. 21
4.2 Domain size and pressure boundary conditions ......................... 22
4.3 Inlet turbulence quantities ............................................. 25
4.4 Overall comparison ..................................................... 26

5 Results for different Reynolds numbers 33
5.1 Numerical uncertainty .................................................. 33
  5.1.1 Iterative errors ....................................................... 33
  5.1.2 Discretization errors ................................................. 34
5.2 Numerical results ....................................................... 35
5.3 Experimental data ....................................................... 37
  5.3.1 Experimental uncertainty ........................................... 38
5.4 Validation exercise ..................................................... 38

6 Conclusions ............................................................... 47

References ................................................................. 51
List of Tables

2.1 Distances of the boundaries to the cylinder centre for the three domain sizes tested in the calculation of the two-dimensional flow around a squared cylinder with rounded corners at $R_{n} = 1.74 \times 10^{5}$. 13

2.2 Distances of the boundaries to the cylinder centre for the two domain sizes tested in the calculation of the two-dimensional flow around a squared cylinder with rounded corners at various Reynolds numbers. 13

2.3 Turbulence quantities specified at the inlet of the domain in the calculation of the two-dimensional flow around a squared cylinder with rounded corners at $R_{n} = 1.74 \times 10^{5}$. 14

3.1 Details of the finest refinement used for the calculations of the two-dimensional flow around a squared cylinder with rounded corners at six Reynolds number with two different sizes of the computational domain. 19

4.1 Data range in percentage of the selected functional flow quantities of calculation of the two-dimensional flow around a squared cylinder with rounded corners at $R_{n} = 1.74 \times 10^{5}$. 23

4.2 Largest differences of average drag coefficient $(C_{D})_{avg}$, maximum $(C_{L})_{max}$ and root mean squared $(C_{L})_{rms}$ lift coefficient and Strouhal number $St$. Calculation of the two-dimensional flow around a squared cylinder with rounded corners at $R_{n} = 1.74 \times 10^{5}$ with different domain sizes, pressure boundary conditions and inlet turbulence quantities. 27

5.1 Data range of the selected functional flow quantities of calculation of the two-dimensional flow around a squared cylinder with rounded corners at two different Reynolds numbers, as function of iterative converge criteria. Results for two different sizes of the computational domain. 34

5.2 Data range in percentage of the selected functional flow quantities of calculation of the two-dimensional flow around a squared cylinder with rounded corners for different Reynolds numbers, $1 \leq r_{i} \leq 1.71$. 35

5.3 Data range in percentage of the selected functional flow quantities of calculation of the two-dimensional flow around a squared cylinder with rounded corners for different Reynolds numbers, $1 \leq r_{i} \leq 1.33$. 36

5.4 Parameters of the refinement level used to calculate flow around squared cylinder of rounded corners for all Reynolds numbers. 36
5.5 Flow quantities of the refinement level used to calculate flow around squared cylinder of rounded corners for all Reynolds numbers. ............................................. 37
5.6 Experimental data for the flow quantities, \( (C_D)_{avg} \) and \( St \). ......................................................... 38
5.7 Validation results for drag coefficient, \( (C_D)_{avg} \). .......................................................... 39
List of Figures

1.1 Vortex shedding parameters as a function of the Reynolds Number for a smooth circular cylinder in a uniform low-turbulence stream: the average drag coefficient (top), root mean square lift coefficient (middle) and Strouhal number (bottom) (source: [23]). ............................. 5

1.2 Vortex shedding parameters as a function of the Reynolds Number for a smooth square cylinder in a uniform low-turbulence stream: the average drag coefficient (top) and Strouhal number (bottom). Single Re-number results marked as ‘others’: Pocha (1971), [25], Lee (1975), [26], Bearman and Obasaju (1982), [27], Obasaju (1983), [28], Sarpkaya and Ihrig (1986), [29], Durao et al. (1988), [30], and Lyn(1992), [31], (source: [23]). ........................................ 7

2.1 Domain for the calculation of the two-dimensional flow around a squared cylinder of rounded corners at \( R_n = 1.74 \times 10^5 \). ...................................................... 12

3.1 Section of the rounded-off square cylinder ............................................. 16

3.2 Overview of grid layout (left) and detail of the mesh close to the cylinder (right) .......... 17

3.3 Illustration of the coarsest grids for the calculation of the two-dimensional flow around a squared cylinder of rounded corners at \( R_n = 1.74 \times 10^5 \). ...................................................... 18

3.4 Illustration of the coarsest grids for the calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds number. ................................. 18

4.1 Illustration of the iterative convergence of the calculation of the two-dimensional flow around a squared cylinder of rounded corners at \( R_n = 1.74 \times 10^5 \). \( N_{\text{iterations}} \) is the number of iterations performed at each time step. \( \phi_{\text{avg}} \) is the value of variable \( \phi \) obtained from the average of the last 8 cycles. Finest grid of DM size with IM inlet conditions and pressure fixed at the top left corner of the domain. .......................................................... 22

4.2 Average drag coefficient \( (C_D)_{\text{avg}} \) as a function of the grid/time refinement ratio (left); average surface pressure coefficient \( (C_p)_{\text{avg}} \) distribution \( (\theta = 0 \text{ at } x = -0.5D, y = 0 \text{ and } \theta = 180^\circ \text{ at } x = 0.5D, y = 0) \) for \( r_i = 1 \) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \( R_n = 1.74 \times 10^5 \) with three different sizes of the computational domain and different pressure boundary conditions. IM inlet conditions. 24
4.3 Maximum lift coefficient \((C_L)_{\text{max}}\) as a function of the grid/time refinement ratio (left); surface pressure coefficient \((C_p)_{\text{avg}}\) distribution \((\theta = 0 \text{ at } x = -0.5D, y = 0, \theta = 180^\circ \text{ at } x = 0.5D, y = 0 \text{ and } \theta = -90^\circ \text{ at } x = 0, y = -0.5D)\) at the time corresponding to \((C_L)_{\text{max}}\) for \(r_i = 1\) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(Rn = 1.74 \times 10^5\) with three different sizes of the computational domain and different pressure boundary conditions. IM inlet conditions.

4.4 Skin friction coefficient \(C_f\) on the cylinder surface: average value (left); value at time corresponding to \((C_L)_{\text{max}}\). \((\theta = 0 \text{ at } x = -0.5D, y = 0, \theta = 180^\circ \text{ at } x = 0.5D, y = 0, \theta = -90^\circ \text{ at } x = 0, y = -0.5D \text{ and } C_f \text{ multiplied by -1 for negative } \theta)\). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(Rn = 1.74 \times 10^5\) with three different sizes of the computational domain and different pressure boundary conditions. IM inlet conditions.

4.5 Streamlines at times corresponding to \((C_L)_{\text{max}}\) (left) and \(C_L = 0\) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(Rn = 1.74 \times 10^5\) with three different sizes of the computational domain and pressure fixed at 1 point. IM inlet conditions.

4.6 Strouhal number \(St\) (left) and root mean squared of the lift coefficient \((C_L)_{\text{rms}}\) as a function of the grid/time refinement ratio (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(Rn = 1.74 \times 10^5\) with three different sizes of the computational domain and different pressure boundary conditions. IM inlet conditions.

4.7 Average drag coefficient \((C_D)_{\text{avg}}\) as a function of the grid/time refinement ratio (left); average surface pressure coefficient \((C_p)_{\text{avg}}\) distribution \((\theta = 0 \text{ at } x = -0.5D, y = 0 \text{ and } \theta = 180^\circ \text{ at } x = 0.5D, y = 0)\) for \(r_i = 1\) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(Rn = 1.74 \times 10^5\) with three different levels of the inlet turbulence quantities. Domain size and pressure boundary conditions given in the plots legend.

4.8 Maximum lift coefficient \((C_L)_{\text{max}}\) as a function of the grid/time refinement ratio (left); surface pressure coefficient \((C_p)_{\text{avg}}\) distribution \((\theta = 0 \text{ at } x = -0.5D, y = 0, \theta = 180^\circ \text{ at } x = 0.5D, y = 0 \text{ and } \theta = -90^\circ \text{ at } x = 0, y = -0.5D)\) at the time corresponding to \((C_L)_{\text{max}}\) for \(r_i = 1\) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(Rn = 1.74 \times 10^5\) with three different levels of the inlet turbulence quantities. Domain size and pressure boundary conditions given in the plots legend.

4.9 Skin friction coefficient \(C_f\) on the cylinder surface: average value (left); value at time corresponding to \((C_L)_{\text{max}}\). \((\theta = 0 \text{ at } x = -0.5D, y = 0, \theta = 180^\circ \text{ at } x = 0.5D, y = 0, \theta = -90^\circ \text{ at } x = 0, y = -0.5D \text{ and } C_f \text{ multiplied by -1 for negative } \theta)\). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(Rn = 1.74 \times 10^5\) with three different levels of the inlet turbulence quantities. Domain size and pressure boundary conditions given in the plots legend.
4.10 Streamlines at times corresponding to \((C_L)_{\text{max}}\) (left) and \(C_L = 0\) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(R_n = 1.74 \times 10^5\) with three different levels of the inlet turbulence quantities. Smallest domain size DS with pressure fixed at 1 point. .................................................. 30

4.11 Eddy-viscosity \((\nu_t)\) isolines at times corresponding to \((C_L)_{\text{max}}\) (left) and \(C_L = 0\) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(R_n = 1.74 \times 10^5\) with three different levels of the inlet turbulence quantities. Smallest domain size DS with pressure fixed at 1 point. .................................................. 30

4.12 Strouhal number \(St\) (left) and root mean squared of the lift coefficient \((C_L)_{\text{rms}}\) as a function of the grid/time refinement ratio (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(R_n = 1.74 \times 10^5\) with three different levels of the inlet turbulence quantities. Domain size and pressure boundary conditions given in the plots legend. .................................................. 31

5.1 Illustration of the iterative convergence of the calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(R_n = 1.74 \times 10^5\). \(N_{\text{iterations}}\) is the number of iterations performed at each time step. Time history of the force coefficients, as a function of iterative converge criteria, \(c_{it}\): \(10^{-3}\) (left) and \(10^{-6}\) (right). Finest grid of DT size domain. .................................................. 40

5.2 Average surface pressure coefficient \((C_p)_{\text{avg}}\) (left) and average skin friction coefficient \((C_f)_{\text{avg}}\) (right) distribution \((\theta = 0)\) at \(x = -0.5D, y = 0\) and \(\theta = 180^\circ\) at \(x = 0.5D, y = 0\) for \(r_i = 1\). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(R_n = 1.74 \times 10^5\) with DT size of the computational domain. Solid lines for \(c_{it} = 10^{-6}\) and dashed lines for \(c_{it} = 10^{-3}\). .................................................. 41

5.3 Time histories of the drag coefficient(red line) and the lift coefficient (blue line) for six values of \(R_n\) with finest refinement level, \(r_i = 1\). .................................................. 42

5.4 Average drag coefficient \((C_D)_{\text{avg}}\) as a function of the grid/time refinement ratio (left); average surface pressure coefficient \((C_p)_{\text{avg}}\) distribution \((\theta = 0)\) at \(x = -0.5D, y = 0\) and \(\theta = 180^\circ\) at \(x = 0.5D, y = 0\) for \(r_i = 1\) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT. Reynolds number given in the plots legend. .................................................. 43

5.5 Maximum lift coefficient \((C_L)_{\text{max}}\) as a function of the grid/time refinement ratio (left); surface pressure coefficient \((C_p)_{\text{avg}}\) distribution \((\theta = 0)\) at \(x = -0.5D, y = 0\), \(\theta = 180^\circ\) at \(x = 0.5D, y = 0\) and \(\theta = -90^\circ\) at \(x = 0, y = -0.5D\) at the time corresponding to \((C_L)_{\text{max}}\) for \(r_i = 1\) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT. Reynolds number given in the plots legend. .................................................. 43
5.6 Skin friction coefficient $C_f$ on the cylinder surface: average value (left); value at time corresponding to $(C_L)_{max}$. ($\theta = 0$ at $x = -0.5D, y = 0$, $\theta = 180^\circ$ at $x = 0.5D, y = 0$, $\theta = -90^\circ$ at $x = 0, y = -0.5D$ and $C_f$ multiplied by -1 for negative $\theta$). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT. Reynolds number given in the plots legend.

5.7 Streamlines at times corresponding to $(C_L)_{max}$ (top) and $C_L = 0$ (bottom). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT. Reynolds number given in the plots legend.

5.8 Strouhal number $St$ (left) and root mean squared of the lift coefficient $(C_L)_{rms}$ as a function of the grid/time refinement ratio (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT. Reynolds number given in the plots legend.

5.9 Influence the domain size at the transition between subcritical and supercritical regime for the flow quantities.
Nomenclature

Acronyms

2D Two-Dimensional
CFD Computational Fluid Dynamics
DL Largest Domain
DM Medium Domain
DNS Direct Numerical Simulation
DS Smallest Domain
DS Towing Tank Domain
IH Higher Inlet turbulence quantities
IL Lower Inlet turbulence quantities
IM Middle Inlet turbulence quantities
LES Large-Eddy Simulation
NS Navier-Stokes equations
PANS Partially-Averaged Navier-Stokes
RANS Reynolds-averaged Navier-Stokes
SAS Scale Adaptive Simulation
SIMPLE Semi-Implicit Method for Pressure-Linked Equations
SST $k - \omega$ Shear Stress Transport $k - \omega$ eddy-viscosity model
URANS Unsteady Reynolds-averaged Navier-Stokes

Greek symbols

$\alpha$ Flow direction angle
$\Delta t$ Time step
κ Turbulent kinetic energy
μ Molecular viscosity
ν Kinematic molecular viscosity
ν_τ Kinematic turbulent (eddy) viscosity
ω Specific turbulent dissipation rate
ϕ, ¯ϕ, ϕ′ Instantaneous, time-averaged and fluctuation flow variable
ρ Density
τ_{ij} Viscous stress tensor
τ_w Wall shear stress
θ Angle of the cylinder surface

Roman symbols

C_D,C_L Drag and lift coefficients
C_f Skin friction coefficient
C_p Pressure coefficient
c_{it} Iterative convergence criteria
D Diameter of the squared cylinder
F Total force exerted on a cylinder (pressure + viscous)
F_D,F_L Drag and lift components of the total force
f_s Vortex shedding frequency
L Length of the squared cylinder
L_∞ Maximum normalized residual norm
L_{rec} Length of the mean recirculation zone
N_{iterations} Number of iterations
p_∞ Undisturbed pressure
r_c Radius of the rounded corners
r_i Refinement ratio
R_n Reynolds number
St Strouhal number
$T$  Averaging time interval

$t$  Time

$T_s$  Vortex shedding period

$u_i, u_j, u_k$  Velocity components in $x, y, z$ directions

$U_\infty$  Undisturbed velocity

$y^+$  Non-dimensional distance from the wall

**Subscripts**

$avg$  Average

$f$  Friction or viscous

$i, j, k$  Unit vectors in $x, y, z$ Cartesian directions

$max$  Maximum

$min$  Minimum

$p$  Pressure

$rms$  Root-mean-square

$w$  Wall

$x, y, z$  Cartesian components

**Superscripts**

$T$  Transpose.
Chapter 1

Introduction

1.1 Motivation

Many practical constructions use cylinders that range from circular cross-sections to square cylinders with rounded corners. The flow around square cylinders with corner modifications has attracted a great deal of attention because of its practical significance in industrial installations. Practical examples are tall buildings, monuments, towers and cables in suspension bridges, hoardings and signs (civil engineering), landing gears and fuselage cross section (aeronautical engineering) which are permanently exposed to wind. Similarly, heat exchanger pipe bundles (process engineering and energy conversion) and piers, bridge pillars and legs of offshore platforms (maritime engineering) are subjected to loads produced by water streams. At the typical Reynolds numbers of practical applications, these bodies create a large region of separated flow that leads to an unsteady wake region where vortex shedding occurs. This type of flow leads to unsteady (periodic) lift and drag forces acting on the cylinders and to fluctuations in the flow field that are related to the shedding process. Therefore, a flow around a square cylinder may originate damaging oscillations when the natural frequency of the obstacle is close to the shedding frequency of the vortices. If the resulting excitation frequency synchronizes with the natural frequency of a cylinder, the phenomenon of resonance is the obvious outcome.

Typical Reynolds numbers of practical applications are in the range of $10^5$ to $10^6$ where the so-called "drag crisis" occurs, [1]. Flow simulations in such conditions are extremely challenging because the flow exhibits laminar, transitional and turbulent regions. Furthermore, due to the existence of vortex shedding, the flow is not statistically steady. Although many of the publications available in the open literature focus on the circular cylinder, there is a growing interest in square shapes with rounded corners [2],[3], [4], [5], [6], [7] and [8].

Recently [9], a squared cylinder with rounded corners of 9% of the diameter has been tested in a towing tank for Reynolds numbers $Rn$ based on the undisturbed incoming flow $U_\infty$ and the cylinder width $D$ ranging from $7.2 \times 10^4$ to $3.1 \times 10^5$. The results obtained for the lift and drag force suggest the existence of different flow characteristics in this range of Reynolds numbers, which is in agreement
with of the studies reported in [2], [3] and [4]. Therefore, it is important to address this problem with mathematical models (instead of physical models) to obtain some insight about the origin of these flow changes.

### 1.2 Flow around cylinders

#### 1.2.1 Forces acting on a cylinder

The force acting on a cylinder immersed in a uniform flow is the sum of a pressure force, $F_p$, and a viscous or friction force, $F_f$:

$$ F = \oint_{cyl} p n dA - \oint_{cyl} \tau_{ij} n_j dA = F_p + F_f \quad (1.1) $$

Here $p$ is the pressure on the cylinder surface, $\tau_{ij}$ is the viscous stress tensor, $dA$ and $n = n_j$ denote differential surface area and its outward unit vector, respectively. The pressure and wall shear-stress $\tau_w$ are normally given as dimensionless quantities through the definition of the pressure coefficient, $C_p$, and skin friction coefficient, $C_f$:

$$ C_p = (p - p_\infty)/(1/2 \rho U_\infty^2), \quad C_f = \tau_w/(1/2 \rho U_\infty^2). \quad (1.2) $$

where $p_\infty$ is the undisturbed pressure.

The total force $F$ is usually decomposed into two components: the drag force $F_D$ aligned with the incoming flow; the lift force $F_L$ perpendicular to the incoming flow. The corresponding non-dimensional forces, namely drag and lift coefficients are defined by:

$$ C_D = F_D/(1/2 \rho U_\infty^2 DL), \quad C_L = F_L/(1/2 \rho U_\infty^2 DL). \quad (1.3) $$

where $L$ is the cylinder length$^1$

Forces generated by the flow around cylinders may be steady or unsteady depending on the Reynolds number. For the unsteady situations (that correspond to most practical applications), these coefficients are normally presented as the sum of a time-averaged and a fluctuating component. For example, the drag coefficient is assembled as:

$$ C_D = \bar{C}_D + C_D', \quad \bar{C}_D = \frac{1}{T} \int_{t_0}^{t_0+T} C_D dt. \quad (1.4) $$

The time-averaged effect of the fluctuating forces is quantified by the root-mean-square (r.m.s.) values of the drag and lift coefficients. For example, the r.m.s. lift coefficient $(C_L)_{rns}$ reads:

$$ (C_L)_{rns} = \left[ \frac{1}{T} \int_{t_0}^{t_0+T} (C_L^2 - \bar{C}_L^2) dt \right]^{1/2}. \quad (1.5) $$

$^1$When the flow is assumed to be two-dimensional $L = 1.$
1.2.2 Vortex shedding

The main feature of the flow around cylinders is the separation of fluid from the surface that generates vortices in the near-wake region. These vortices may remain attached to the body or be convected to the far wake periodically leading to the well known phenomena of vortex shedding. The different regimes of the flow are controlled by the Reynolds number, \( Rn \). For a cylinder of reference length \( D \) (diameter or width) placed in a uniform flow of velocity, \( U_\infty \), the Reynolds number is defined by:

\[
Rn = \frac{\rho U_\infty D}{\mu} .
\]  
(1.6)

At the typical Reynolds numbers of practical applications, the near-wake presents always vortex shedding which is characterized by a shedding frequency \( f_s \) that is usually quantified by the Strouhal number, which is defined by:

\[
St = \frac{f_s D}{U_\infty} .
\]  
(1.7)

The power spectra of the time-dependent force coefficients or velocity components in the wake can be used to determine the frequency of the shedding process. In the literature, see for example [1], the vortex shedding frequency, \( f_s \), is generally associated with the dominant frequency in the power spectra of the lift coefficient \( C_L \) fluctuations.

Circular cylinder

For cylinder with circular cross-sections, the flow regimes were first defined by Roshko (1954) (see Williamson (1996), [10]) and recently by Williamson (1996), [10]. Coutanceau and Defaye (1991), [11], gave an extensive description of the flow including the near and far wake based on the flow visualizations. In Figure 1.1, the results from different sources are assembled to show the variation of \((C_D)_{avg}\), \((C_L)_{rms}\) and \(St\) with the Reynolds number. Note that data from other sources, obtained for \( Rn \) beyond \( 10^4 \), are considerably more scattered. This is evident when comparing the present figures with ones compiled by Lienhard (1966), [12], Ericsson (1980), [13], James et al. (1980), [14], Cantwell and Coles (1983), [15], - all for \((C_D)_{avg}\) and \(St\), and by Basu (1985), [16], and West and Apelt(1993), [17], for \((C_L)_{rms}\). In practice, the lift forces can be measured either over the whole span of a cylinder (Schewe, 1983, [18]) or only on the mid-span section (e.g. West and Apelt, 1993, [17]). Different results will be obtained if there is no perfect correlation between the fluctuating forces along the span, Figure 1.1 (middle).

The following flow regimes can be identified (with reference to the labels shown on Figure 1.1):

- **Laminar steady flow**: \( Rn < 40 \) (regime up to A). As \( Rn \) increases, the drag coefficient decreases.

- **Unstable wake and laminar vortex shedding**: \( Rn \approx 40 – 190 \) (regime A-B). The drag coefficient continues to decrease, while the Strouhal number shows a strong increase.
• **Wake transitions**: $Rn \approx 190 - 1000$ (regime B-C). The wake becomes three-dimensional and undergoes a transition to a turbulent regime.

• **Shear layer transition regime**: $Rn \approx 1000 - 3 \times 10^5$ (regime C-D). A transition to turbulence in the shear layers is the main feature of this subcritical regime. At $Rn \approx 10^4$ the transition occurs close to the separation point. However, it is preceded by laminar separation of the boundary layers. For $Rn > 10^4$, the separation point has a quasi-constant value of $80^\circ$ (Achenbach, 1968, [19]; Britter et al., 1979, [20]). The fairly constant values of $(C_D)_{avg} = 1.0 - 1.2$, and $St \approx 0.20$ correspond with a turbulent motion in the vortex street and in the major part of the shear layers.

• **Asymmetric reattachment** (critical regime): $Rn \approx (3 - 4) \times 10^5$ (regime D-E). The transition from laminar to turbulent flow occurs just downstream of the laminar separation point ($\theta_s = 80^\circ - 100^\circ$). The critical flow regime appears when one of the shear layers reattaches to the cylinder wall, forming a separation bubble. The revitalized turbulent boundary layer separates again, only further downstream ($\theta_s = 120^\circ - 140^\circ$; Achenbach, 1968, [19]). This is followed by a significant decrease of the wake width and drag coefficient (the ‘drag crisis’), and by an increase of $St$. In this transitional regime, the flow is extremely sensitive to three-dimensional disturbances, free-stream turbulence and surface roughness and so vortex shedding is not regular.

• **Symmetric reattachment** (supercritical regime): $Rn \approx 4 \times 10^5 - 10^6$ (regime E-F). The mean symmetric flow, with two separation-reattachment bubbles, and nearly constant minimal $(C_D)_{avg} \approx 0.22$ and maximal $St \approx 0.47$ is characterized by approximately regular vortex shedding in the supercritical flow regime.

• **Boundary layer transition** (postcritical regime): $Re > 10^6$ (regime from F). At $Rn > 10^6$, the boundary layers become turbulent before separating at $\theta_s \approx (110 - 120)^\circ$. For $Rn > 5 \times 10^6$, the dominant frequency $St \approx 0.27 - 0.29$ indicates a strong vortex shedding process (see also Roshko, 1961 and Jones et al., 1969). The drag coefficient appears to be nearly constant $(C_D)_{avg} \approx 0.5$ over this postcritical regime.

**Square cylinder**

In contrast to circular cylinders, bluff bodies with fixed separation points, like cylinders with rectangular cross-sections, have attracted much less attention. The experimental results available for a square cylinder are presented in Figure 1.2. Similarity of the flow changes with the $Rn$ with those described above for a circular cylinder are obvious. However, the flow does not exhibit the remarkable changes associated with the transition phenomenon in the critical regime that are observed for the circular cylinder. According to Okajima (1982), [21], the flow quantities are generally less sensitive to changes in the $Rn$ if the flow separates at fixed points. At low $Rn$, $Rn < 150$, Franke (1991), [22], the separation points are fixed at the rear corners of the square cylinder. At higher $Rn$, the separation happens at the front corners, and as for the circular cylinder, a transition from laminar to turbulent vortex street can be
expected. For example, the power spectra of the wake velocities measured by Okajima (1982), [21], showed secondary frequencies at $Rn = 250$. Beyond $Rn = 10^4$, the mean drag coefficient and Strouhal number attain nearly constant values ($(C_D)_{avg} = 2 - 2.2$ and $St = 0.12 - 0.14$).

Figure 1.1: Vortex shedding parameters as a function of the Reynolds Number for a smooth circular cylinder in a uniform low-turbulence stream: the average drag coefficient (top), root mean square lift coefficient (middle) and Strouhal number (bottom) (source: [23]).

Square cylinder with rounded corners

Several investigations have been conducted to address the effect of corner modifications on the flow characteristics of square cylinders. The pioneering studies conducted by Delany and Soresen (1953), [5] showed a drop in drag coefficient around the Reynolds Number of $6.5 \times 10^5$ or $2.5 \times 10^5$ for a square cylinder with two different rounded corners, $(r_c/D = 0.167)$ and $(r_c/D = 0.333)$, respectively. Tamura et
al. (1998, [6], 1999, [7]) conducted wind tunnel tests to investigate the effect of the corner modification on the aerodynamic forces of two-dimensional (2D) and three-dimensional square cylinders in smooth and turbulent flow. A numerical simulation was also conducted for smooth surface. The results showed that chamfered and rounded corners decrease drag force because of the reduction of wake width. The mechanisms of aerodynamic force reduction were also clarified by the numerical simulation results. Furthermore, it was found that the shear layer that is separated from the leading edge reattached to the side surfaces of the corner-rounded cylinder in turbulent flow.

Wind tunnel tests conducted by Carassale et al. (2012, [2], 2013, [4], 2014, [3]) were conducted to measure the aerodynamic forces and the wind pressures of square cylinders with sharp-edge corner and two different rounded corner ratios \( r_c/D = 1/15 \) and \( r_c/D = 2/15 \) at Reynolds numbers ranging between \( 1.7 \times 10^4 \) and \( 2.3 \times 10^5 \). The tests included different angles between the cylinders and the incoming flow. The rounded corner results showed a reduction in the critical angle of incidence for which the flow reattaches to the lateral face exposed to wind. Moreover, the Reynolds number effects on the square cylinder with the rounded corner ratio \( r_c/D = 2/15 \) are observed because of the presence of transition phenomena involving flow reattachment on the lateral faces and the potentially unstable range of angle shifts with the Reynolds number in turbulent flow. As previously demonstrated, the aerodynamic characteristics of the square cylinder may drastically change with slight changes in rounded corner shape, as seen by Letchford and Mason (2011), [8]. The rounded corners may cause the absence of the fixed flow separation point, which influences the formation of the separated shear layer at the leading edge. Thus, the presence or the extent of the flow reattachment regions may also be affected. This brief review shows that previous research mainly aimed at the effects of modified corners, such as rounded corners, chamfered corners, and concave corners, on the aerodynamic forces for square cylinders. Studies on the Reynolds number effects on square cylinder with various corner shapes are relatively few, but have attracted increasing attention in recent years.

Finally, it should be mentioned that vortex shedding from circular and square cylinders with or without rounded corners is also affected by the surface roughness, the free stream turbulence, the cylinder aspect ratio and wind tunnel blockage. The influence of these parameters was reviewed by Basu (1985, [16], 1986 [?]), Bell (1986), [24], Coutanceau and Defaye (1991), [11], and Williamson (1996), [10].

1.3 Objectives

The main objective of this thesis is to study the flow around a squared cylinder with rounded corners using mathematical modelling, i.e. flow simulations. The selected geometry was tested in a towing tank and average drag coefficients are reported in [9] suggesting the existing of different flow regimes for the range of Reynolds numbers tested. The main goal of this work is to investigate the ability to capture the dependency of the flow regimes on the Reynolds number. However, in order to assess the quality of the mathematical modelling, it is also important to address the influence of the different components of the model on the outcome of the simulations, i.e. the domain size, mass and momentum balance modelled
To this end we have chosen the simplest model to simulate high Reynolds numbers wall-bounded turbulent flows: the Reynolds-Averaged (ensemble average) continuity and Navier-Stokes (RANS) equations supplemented by the eddy-viscosity two-equation SST $k - \omega$ model [32]. Furthermore, we have assumed two-dimensional flow. On the other hand, several sensitivity studies are conducted to determine the influence of the remaining components of the mathematical model, namely:

- The size of the domain.
- The specification of the inlet turbulence quantities.
- The specification of the pressure boundary conditions.

For all these sensitivity test, grid/time refinement studies are performed to estimate the numerical uncertainty of the solutions [33], which is a fundamental component of any proper Validation procedure.
[34], i.e. of the estimation of the modelling error.

Although we are aware of the shortcomings of the selected model for such complex flows, the studies described above are extremely time-consuming, even for this model. Furthermore, the problems addressed in this study are common to more sophisticated models that would make this type of investigation unaffordable.

To fulfill the main goal of this work, the flow around the selected geometry is calculated for the Reynolds numbers used in the experiments reported in [9]. In order to assess the influence of the problem settings, two different domains were selected:

- A domain that corresponds to the dimensions of the towing tank.
- A sufficiently large domain to have a negligible effect of the selected pressure boundary conditions.

Finally, the ASME V&V 20 Validation procedure [34] is applied to the results obtained in the domain corresponding to the towing tank to evaluate the modelling error of the two-dimensional RANS equations supplemented by the SST $k - \omega$ model [32]. This exercise also illustrates the requirements (problem settings, experimental and numerical uncertainties,...) for a proper assessment of the modelling error.

### 1.4 Structure of the thesis

The outline of this document is as following:

- Chapter 1 (this chapter) gives an introduction to flow around cylinders and the previous findings by other authors.
- Chapter 2 outlines the mathematical model domain size and boundary conditions used for the simulations.
- Chapter 3 presents the numerical solution, namely the ReFRESCO flow solver, the geometry of test case and the mesh, including a time/grid convergence study.
- Chapter 4 presents and discusses the results obtained at the sensitivity study for the domain size, the inlet turbulence quantities and the pressure boundary condition from the simulation of the around square cylinder with rounded corners.
- Chapter 5 presents and discusses the results from the simulation of the various Reynolds numbers and gives a validation of the numerical model.
- Chapter 6 finally summarises and concludes the findings and give recommendations for further work.
Chapter 2

Mathematical Model

The Navier-Stokes equations are a cornerstone of fluid mechanics. The equations describe viscous fluid flows. In most cases, even when simplifying the Navier-Stokes equations, they are not possible to be solved analytically. The solution is to use numerical methods.

2.1 Flow regime

The Navier-Stokes equations are partial differential equations that describe the flow of viscous fluids. For Newtonian fluids, the mass conservation equation and the momentum balance equations written in Cartesian coordinates and in an inertial space-fixed reference frame, see for instance [35], in differential form,

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0, \]

\[ \frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_j u_i)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \tau_{ij}, \]

(2.1)

where \( u_i \) represents the i-the component of the fluid velocity at a point in space, \( x_i \), and time, \( t \). Also \( p \) is the pressure relative to the hydrostatic pressure, i.e. if gravity is the only body force, \( \rho \) is the fluid density and \( \tau_{ij} \) is the viscous stress tensor for Newtonian fluid, can be written:

\[ \tau_{ij} = 2\mu S_{ij}^* \]

(2.2)

with

\[ S_{ij}^* = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \]

(2.3)

where \( \mu \) is the molecular viscosity.

2.1.1 Reynolds-Averaged Navier-Stokes equations

The present model is intended to deal with turbulent flows. Turbulent flows are highly unsteady and several time and length scales are usually presented in realistic turbulent flows.
In a statistically steady flow, every variable $\phi$ can be written as the sum of a time-averaged, $\bar{\phi}$, plus a fluctuation, $\phi'$, about that value, see for instance [35],

$$\phi(x, t) = \bar{\phi}(x_i) + \phi'(x_i, t),$$  \hspace{1cm} (2.4)

where

$$\bar{\phi}(x_i) = \lim_{T \to \infty} \frac{1}{T} \int_{t_0}^{t_0+T} \phi(x_i, t) \, dt.$$  \hspace{1cm} (2.5)

Here $t$ is the time and $T$ is the averaging interval. In the flow is unsteady, time averaging cannot be used and it must be replaced by ensemble averaging,

$$\bar{\phi}(x_i) = \lim_{N \to \infty} \sum_{n=1}^{N} \phi(x_i, t_n), \quad t_n = t_0 + n\Delta t,$$  \hspace{1cm} (2.6)

where $N$ is the number of the members of the ensemble and must be large enough to eliminate the effects of the fluctuations. This type of averaging can be to applied to any flow. We denote both these averaging processes by Reynolds averaging and if we apply them to the Navier-Stokes equations presented before, Eq. (2.1), we get the Reynolds-Averaged Navier-Stokes equations, usually denoted by RANS equations. From Eq. (2.5), it follows that $\bar{\phi}' = 0$. Thus, averaging any linear term in the conservation equations simply gives the identical term for the averaged quantity. For a quadratic non-linear term we get two terms, the product of the average and a covariance:

$$u_{i}\phi = (U_i + u'_i)(\bar{\phi} + \phi'),$$  \hspace{1cm} (2.7)

where for the velocity components $u_i$ we consider that

$$u_i(x_i, t) = U_i(x_i, t) + u'_i(x_i, t).$$  \hspace{1cm} (2.8)

The last term is zero only if the two quantities are uncorrelated, this is rarely the case in turbulent flows and, as a result, the conservation equations contain terms such as $\rho u'_i u'_j$, called the Reynolds stress, and $u'_i \phi'$, known as the turbulent scalar flux, among others. These terms cannot be represented uniquely by mean quantities and have therefore additionally modelled.

The averaged continuity and momentum equations can, for incompressible flows, be written in tensor notation and Cartesian coordinates as:

$$\frac{\partial U_i}{\partial x_i} = 0,$$  \hspace{1cm} (2.9)

$$\frac{\partial U_i}{\partial t} + \frac{\partial}{\partial x_j} \left( U_i U_j + u'_i u'_j \right) = -\frac{1}{\rho} \left( \frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} \right),$$

where the $\tau_{ij} = \tau_{ji}$ are the mean viscous stress tensor components, being $i = j = 1, 2, 3$ running indices:

$$\tau_{ij} = 2 \mu S_{ij} = \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right).$$  \hspace{1cm} (2.10)
In order to compute all mean-flow properties of the turbulent flow, Eq. (2.9), we need a prescription for computing the Reynolds stress tensor, i.e. to close the system of equations we have to introduce equations for the Reynolds stress components. Noting that $\tau_{ij}$ is a symmetric tensor we have 6 additional unknowns. The function of the turbulence modes is to provide these extra equations.

2.1.2 Turbulence model

In laminar flows, energy dissipation, transport of mass, momentum and energy normal to the streamlines are mediated by the viscosity, so it is natural to assume that the effect of turbulence can be represented as an increased viscosity. This leads to the eddy-viscosity model (also called Boussinesq approximation) for the Reynolds stress tensor,

$$-u'_i u'_j = \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij},$$

(2.11)

where the $k$ term is the turbulence kinetic energy given by $k = \frac{1}{2} u'_i u'_j$, and $\nu_t = \frac{\omega}{\rho}$ is the turbulent eddy viscosity. This model assumes that the $\nu_t$ is approximately isotropic, an important simplification which may not be realistic for several flows.

In the present work, the turbulence model is the original version of the $k - \omega$ SST turbulence model by Menter (in 1994), see for instance [32]. This two-equations eddy-viscosity turbulence model solves one transport equation for the turbulent kinetic energy, $k$, and one transport equation for the dissipation per unit kinetic energy, $\omega$, also regarded as the turbulent frequency scale. The transport equation of $k$ and $\omega$, for incompressible flows, may be written as:

\[
\begin{align*}
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} &= P_k - \beta^* \omega k + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right], \\
\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} &= \frac{\gamma}{\nu_t} P \omega - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + F_2 \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j},
\end{align*}
\]

(2.12)

with

\[
\begin{align*}
P_k &= \frac{\partial U_i}{\partial x_j} \left[ \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \right], \\
P_\omega &= \frac{\partial U_i}{\partial x_j} \left[ \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \right],
\end{align*}
\]

(2.13)

In the shear-stress transport version, $\nu_t$ is calculated from

$$\nu_t = \frac{a_1 k}{\max (a_1 \omega, F_2 \Omega)},$$

(2.14)

$a_1 = 0.31$ and

$$F_2 = \tanh \left( \arg_2 \right),$$

$$\arg_2 = \max \left( \frac{2 \sqrt{k}}{0.09 \omega y}, \frac{500 \nu}{\omega y^2} \right),$$

(2.15)

where $y$ is the distance from the field point to the nearest wall and $\Omega = \sqrt{2 \Omega_{ij} \Omega_{ij}}$ is the vorticity magni-
Figure 2.1: Domain for the calculation of the two-dimensional flow around a squared cylinder of rounded corners at $R_n = 1.74 \times 10^5$.

tude, with $\Omega_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$. The constants depend on a blending function, $F_1$,

$$F_1 = \tanh (arg^4) \quad (2.16)$$

with

$$arg = \min \left[ \max \left( \frac{\sqrt{K}}{0.009 \omega y}, \frac{500 \nu}{\omega y^2} \right), \frac{4k}{(\sigma_\omega)_2 C D_{k\omega} y^2} \right] \quad (2.17)$$

and

$$C D_{k\omega} = \max \left( 2 \frac{1}{(\sigma_\omega)_2} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-20} \right) \quad (2.18)$$

The model constants are obtained from:

$$\gamma = F_1 \gamma_1 + (1 - F_1) \gamma_2 \quad , \quad \beta = F_1 \beta_1 + (1 - F_1) \beta_2 \quad , \quad \beta^* = 0.09$$

$$\sigma_k = F_1 (\sigma_k)_1 + (1 - F_1) (\sigma_k)_2 \quad , \quad \sigma_\omega = F_1 (\sigma_\omega)_1 + (1 - F_1) (\sigma_\omega)_2 \quad , \quad F_\omega = 2 (1 - F_1) (\sigma_\omega)_2^{-1}$$

$$\gamma_1 = 0.5532 \quad , \quad \beta_1 = 0.075 \quad , \quad (\sigma_k)_1 = 2 \quad \gamma_2 = 0.4404 \quad , \quad \beta_2 = 0.0828 \quad , \quad (\sigma_k)_2 = 1 \quad \sigma_\omega)_1 = 2 \quad , \quad (\sigma_\omega)_2 = 1.17 \quad (2.19)$$

### 2.2 Domain size

The domain for the calculation of the two-dimensional flow around the rounded corner ($r_c = 0.09D$) cylinder at a Reynolds number of $1.74 \times 10^5$ is a rectangle of variable size that is illustrated in figure 2.1. The inlet and outlet boundaries are $x = const.$ planes located $L_{in}$ and $L_{out}$ upstream and downstream of the cylinder centre, respectively. The external boundaries are $y = const.$ planes located $L_{ext}$ away from the cylinder centre. Three different domains were tested in the sensitivity studies. The corresponding values of $L_{in}$, $L_{out}$, and $L_{ext}$ are given in table 2.1.
Table 2.1: Distances of the boundaries to the cylinder centre for the three domain sizes tested in the calculation of the two-dimensional flow around a squared cylinder with rounded corners at $R_t = 1.74 \times 10^5$.

<table>
<thead>
<tr>
<th>Domain</th>
<th>$L_{in}$</th>
<th>$L_{out}$</th>
<th>$L_{ext}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>DS</td>
<td>5</td>
<td>20</td>
<td>10</td>
</tr>
<tr>
<td>DM</td>
<td>10</td>
<td>40</td>
<td>20</td>
</tr>
<tr>
<td>DL</td>
<td>20</td>
<td>80</td>
<td>40</td>
</tr>
</tbody>
</table>

Table 2.2: Distances of the boundaries to the cylinder centre for the two domain sizes tested in the calculation of the two-dimensional flow around a squared cylinder with rounded corners at various Reynolds numbers.

The unsteady turbulent flow over square cylinder with rounded corners was computed at various Reynolds numbers, ranging from $7.24 \times 10^4$ to $3.13 \times 10^5$. The calculations were computed with two different sizes of the computational domain in the present exercise, the corresponding values of $L_{in}$, $L_{out}$ and $L_{ext}$ are given in table 2.2. The two domains DL and DT are used to investigate the blockage effect in the localization at where “drag crisis” occurs, in the last the selection of this domain has been guided by the towing tank size observed in the experiments of J. H. Helder and F. Tap (2011), [9].

2.3 Boundary conditions

Several alternatives were tested for the specification of the boundary conditions. Nonetheless, there are boundary conditions that remained fixed for all calculations:

- No-slip and impermeability conditions are applied at the cylinder surface (no wall-functions) and the normal pressure derivative is set equal to zero. The turbulence kinetic energy $k$ is set equal to zero and $\omega$ is specified at the near-wall cell centre [33].

- Zero stream wise derivatives are assumed for all flow variables at the outlet boundary.

- At the external boundary, zero normal derivatives have been applied to the $x$ velocity component $u_x$ and to the turbulence quantities $k$ and $\omega$, whereas the $y$ velocity component $u_y$ is set equal to zero.

- At the inlet boundary, $u_x = U_\infty$, $u_y = 0$ and the pressure is extrapolated from the interior assuming zero stream wise derivative.

Two different alternatives were tested for the pressure at the external boundary:

1. Pressure equal to undisturbed flow pressure $p_\infty$, i.e. $C_p = 0$.

2. Zero normal pressure derivative and $C_p = 0$ at the top left corner of the domain.
Table 2.3: Turbulence quantities specified at the inlet of the domain in the calculation of the two-dimensional flow around a squared cylinder with rounded corners at $R_n = 1.74 \times 10^5$.

<table>
<thead>
<tr>
<th>Inlet</th>
<th>$k/U_\infty^2$</th>
<th>$\omega D/U_\infty$</th>
<th>$\nu_t/\nu$</th>
<th>I(%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>IL</td>
<td>$1.5 \times 10^{-4}$</td>
<td>2609.6</td>
<td>0.01</td>
<td>1</td>
</tr>
<tr>
<td>IM</td>
<td>$3.75 \times 10^{-3}$</td>
<td>6525</td>
<td>0.1</td>
<td>5</td>
</tr>
<tr>
<td>IH</td>
<td>$1.5 \times 10^{-2}$</td>
<td>2609.6</td>
<td>1</td>
<td>10</td>
</tr>
</tbody>
</table>

Three different values were tested for the turbulence quantities at the inlet boundary. The selected values of $k$, $\omega$ and the corresponding values of eddy-viscosity $\nu_t$ and turbulence intensity $I$ are given in table 2.2.

It should be mentioned that the $k$ and $\omega$ transport equations [32] in an incoming uniform flow reduce to

$$
\frac{\partial k}{\partial x} = -\beta^* k \omega \quad \Rightarrow \quad k = k_{in} (1 + \beta (x - x_{in}) \omega_{in})^{-\beta^*/\beta} \\
\frac{\partial \omega}{\partial x} = -\beta \omega^2 \quad \Rightarrow \quad \omega = \omega_{in} (1 + \beta (x - x_{in}) \omega_{in})^{-1} ,
$$

(2.20)

where all quantities have been made dimensionless using $D$ and $U_\infty$ as the reference length and velocity scales, respectively. The subscript $in$ designates values at the inlet boundary. The eddy-viscosity in the inflow region is given by

$$
\nu_t = (\nu_t)_{in} (1 + \beta (x - x_{in}) \omega_{in})^{1-\beta^*/\beta} ,
$$

(2.21)

For the SST $k - \omega$ model in the outer flow region [32] $\beta^* = 0.09$ and $\beta = 0.0828$ and so equations (3.2) and (3.3) give a rapid decay of $k$ and $\omega$ in the inlet region, but the value of $\nu_t$ decays much slower than $k$ and $\omega$. For example, for the inlet turbulence quantities given in table 2.2, 2D downstream of the inlet $k$ is less than 1% of $k_{in}$, whereas $\nu_t$ is close to 60% of the inlet value.

The boundary conditions used for all the simulations for various Reynolds numbers are the same as described above. In the case of calculations elaborated in the largest domain, the pressure boundary condition was tested the pressure equal to undisturbed flow pressure. As stated below, the influence on the flow quantities to be reduced and provides benefits to the level of the iterative convergence. On the other domain size, the pressure boundary condition was tested the zero normal pressure derivative and $C_p = 0$ at the top left corner of the domain. The evaluation of the influence of the blockage effect for the high Reynolds numbers of turbulent flow was performed for smaller turbulence quantities, IL.

### 2.4 Quantities of interest

The influence of the domain size and boundary conditions specification is evaluated for the following flow quantities: mean drag coefficient $(C_D)_{avg}$, maximum $(C_L)_{max}$, root mean squared $(C_L)_{rms}$ of the lift coefficient, root mean squared $(C_D)_{rms}$ of the drag coefficient, Strouhal number $St$ and length of the mean recirculation $L_{rec}$. Besides these functional quantities, we will also analyze the surface pressure $C_p = (p - p_\infty)/(1/2 \rho U_\infty^2)$ and skin friction $C_f = \tau_w/(1/2 \rho U_\infty^2)$ coefficients, where $p_\infty$ is the undisturbed pressure and $\tau_w$ is the shear-stress at the wall for the sensitivity studies.
Chapter 3

Numerical Solution

3.1 ReFRESCO flow solver

ReFRESCO is a viscous-flow CFD code that solves multiphase (unsteady) incompressible flows using the Navier-Stokes equations, complemented with turbulence models, cavitation models and volume-fraction transport equations for different phases. The equations are discretized using a finite-volume approach with cell-centered collocated variables, in strong-conservation form, and a pressure-correction equation based on the SIMPLE algorithm is used to ensure mass conservation. Time integration is performed implicitly with first or second-order backward schemes. At each implicit time step, the non-linear system for velocity and pressure is linearized with Picard’s method and either a segregated or coupled approach is used. A segregated approach is always adopted for the solution of all other transport equations. The implementation is face-based, which permits grids with elements consisting of an arbitrary number of faces (hexahedrals, tetrahedrals, prisms, pyramids, etc.), and if needed h-refinement (hanging nodes). State-of-the-art CFD features such as moving, sliding and deforming grids, as well automatic grid refinement are also available. For turbulence modelling, RANS/URANS, SAS and DES approaches can be used (PANS and LES are being currently studied). The code is parallelized using MPI and sub-domain decomposition, and runs on Linux workstations and HPC clusters. ReFRESCO is currently being developed and verified at MARIN (in the Netherlands) in collaboration with IST (in Portugal), USP-TPN (University of Sao Paulo, Brazil), TUDelft (Technical University of Delft, the Netherlands), UoS (University of Southampton, UK) and recently UTwente (University of Twente, the Netherlands) and Chalmers (Chalmers University, Sweden) [36].

3.2 Test case

3.2.1 Geometry

The smooth fixed square cylinder simulated presents diameter, $D$, and the corner radius of the cylinder was chosen to be $r_c = 0.09D$. This choice was made to be able to compare the results of the com-
putations with experimental data available at MARIN. The rounded-off square cylinder was designed to represent a typical column of a semi-submersible.

![Figure 3.1: Section of the rounded-off square cylinder](image)

### 3.2.2 Grid generation

The numerical calculation of the flow around complex geometries requires the use of "good quality" grids. The concept of a good quality grid is clearly dependent on the numerical technique applied and on the type of equations solved. Creating a mesh for high Reynolds number flows around rounded-off square cylinder is a complex process, because the high pressure gradients are often observed close to walls and especially close to the corners of the geometry, which has necessitated a large discretization of the geometry, i.e. a high number of cells along the cylinder surface. In the areas where the pressure gradients are smaller, coarser cells can be used to keep the total number of the cells as low as possible without affecting the solution.

The grids for the present study are multi-block structured, containing five blocks. The block-topology used for building the mesh around a square cylinder is shown in figure 3.2. An O-type grid is used around cylinder, covering the zones close to the body (boundary layer) and close to the body wake (shear-layer zones). The grids are defined by function the number of the cells in the circumferential direction per quadrant of the cylinder. The central block is constituted by the two sub-blocks, wherein on inner block, the mesh is orthogonal and the cells have constant size along the wall, and on outer block, the mesh is generated with an elliptical grid generator with control functions based on the Grape approach, see for instance [37], being that the cells along the outer boundary have uniform size, as illustrated in figure 3.2. And the others blocks, the interior grid also is generated with a elliptical grid generator. The grid node distribution in the normal direction the wall follows a one-sided stretching functions, see for instance [38] are applied to ensure the required $y^+$ for the correct application of the no-slip condition, in case the grid node distribution along the cylinder wall is equally spaced. In the $y$-direction the mesh is symmetric with respect to $y = 0$. For this study, sets of geometrically similar mesh to the larger domain sizes are generated starting from the meshes with smaller domain size, i.e. it is to extend the mesh, and the mesh coincides in zones in are common to the various sizes of domains.

No wall-functions are used (the cylinder is smooth and therefore no roughness, nor rough-wall-
functions are considered), follows the suggestion of [39], and very fine grid is used close to the body surface to maintain (for all grids) a $y^+ \leq 1$ for wall adjacent grid cells, assuring that first nodes were well inside the viscous sub-layer, thus dispensing wall functions. Away from the cylinder the grid is coarse, again to save computational resources.

The grid presented in figure 3.2 has 60 cells per quadrant, i.e. 240 cells along the cylinder wall. However, for each domain size, the coarsest grid were used for calculations has 720 cells along the cylinder surface, whereas the finest grid has 1440 cells. The grid parameter is $h = 1/\sqrt{N_{\text{cells}}}$, since the problem is two-dimensional, because the $z$ components of the Navier-Stokes equations are not solved. Therefore, the grid is identified by grid refinement, $r_x$, is defined by

$$r_x^i = \frac{h_i}{h_1} = \left( \frac{(N_{\text{cells}})_i}{(N_{\text{cells}})_1} \right)^{\frac{1}{2}} \quad (3.1)$$

in which $(N_{\text{cells}})_i$ is the number of the cells of grid used in the calculation to be analysed and $(N_{\text{cells}})_1$ is the finest number of the cells of grid used in the calculations. Once again, the finest grid is identified by $r_x^i = h_i/h_1 = 1$, and the others, larger than one.

The grids are utilised for sensitivity studies about the boundary conditions were different of it is used for the simulation of the flow around a squared cylinder of rounded corners at Reynolds number range between $10^4$ and $10^6$. The grids were generated for the simulation of the flow around a squared cylinder with rounded corners at various Reynolds numbers, the external boundary of the central block is different of defined previously, because for the future work, the calculations are extended to angles of incidence between $0^\circ$ and $45^\circ$, which facilitates the process of the grid generation.

### 3.3 Grid sets and time step

Three sets (one for each domain size) of five geometrically similar grids were generated for the simulation of the flow around a squared cylinder of rounded corners. As illustrated in figure 3.4, in the common part of the computational domains, DS, DM and DL, the grids are coincident and all the sets cover a grid
refinement ratio \( r_i = h_i/h_1 = 2 \). The finest grid of the DS domain contains 254016 cells with 1440 cells on the cylinder surface. The maximum dimensionless distance to the wall of the near-wall cells in the coarsest grid is \((y_2^+)^{\text{max}} < 0.3\).

The time step of each calculation is tuned to obtain an average Courant number close to 3 that corresponds to a maximum Courant number close to 23. The time step \( \Delta t_i \) is refined with the same ratio of the grid refinement, i.e. \( r_i = \Delta t_i/\Delta t_1 = h_i/h_1 \). The dimensionless time step of the finest grids calculation is \( \Delta t U_\infty/D = 0.02083 \) (roughly 335 time steps per period).

For each Reynolds number, two sets (one for each domain size) of the five geometrically similar were generated for the simulation of the flow around a square cylinder of rounded corners. As illustrated in figure 5.1, in the common part of the computational domains, the grids are coincident around the cylinder, covering the zone close to the body (boundary-layer) and close to the body wake (shear-layer zones). The maximum dimensionless distance to the wall of the near-wall cells in the coarsest grid is \((y_2^+)^{\text{max}} < 0.3\).

The time step of each calculation is tuned to obtain an average Courant number close to 5 that...

Figure 3.3: Illustration of the coarsest grids for the calculation of the two-dimensional flow around a squared cylinder of rounded corners at \( Re = 1.74 \times 10^5 \).

Figure 3.4: Illustration of the coarsest grids for the calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds number.
Table 3.1: Details of the finest refinement used for the calculations of the two-dimensional flow around a squared cylinder with rounded corners at six Reynolds number with two different sizes of the computational domain.

The selected discretization schemes are all second-order accurate including the convective terms of the \( k \) and \( \omega \) transport equations. However, flux limiters are applied to all second-order upwind schemes that are used to discretize the convective terms of all equations solved. The segregated solver is used for the solution of the momentum and continuity equations at each time step.

All the calculations were performed in double precision to guarantee that the round-off error is negligible when compared to the discretization error. The iterative convergence criteria applied at each time step of the last 200 dimensionless time units requires a maximum normalized residual of all equations solved (momentum balance, mass conservation and \( k \) and \( \omega \) transport equations) below \( \text{Res}_{\text{max}} < 10^{-6} \). The normalized residuals are equivalent to the variables change in a simple Jacobi iteration. However, to facilitate the start of the vortex shedding, all the simulations are started with 20 dimensionless time units calculated with an iterative convergence criteria of \( 10^{-2} \).

For each Reynolds number, two different levels of \( c_{it} \) were tested to evaluate the influence of the iterative error: \( 10^{-3} \) and \( 10^{-6} \). However, the main calculations have used \( c_{it} = 10^{-6} \). All calculations are initialized with 200 dimensionless time units calculated with an iterative convergence criteria of \( 10^{-2} \), to facilitate the start of the vortex shedding, with IH inlet conditions, and at least 200 dimensionless time units are calculated for each level iterative convergence. The influence of iterative error was determined for the periodic solution, i.e., when the influence of the initial condition is reduced to negligible levels.
Chapter 4

Sensitivity studies

4.1 Numerical uncertainty

4.1.1 Iterative errors

Figure 4.1 illustrates the typical iterative convergence results with the data obtained for the finest grid of the DM domain, the inlet conditions IM and the pressure fixed at the top left corner of the domain. The two top plots present the $L_2$ and $L_\infty$ norms of the final normalized residuals of all equations solved at each time step. For the present convergence criteria ($L_\infty$ norm smaller than $10^{-6}$), the $L_2$ norm of all normalized residuals is at least one order magnitude below the accepted tolerance. These two pictures also include the number of iterations ($N_{\text{iterations}}$) performed at each time, which remains between 80 and 160 iterations when the solution becomes periodic.

The two bottom plots of figure 4.1 demonstrate that 200 dimensionless time units are sufficient to obtain a solution with a negligible influence of the initial condition. However, we should emphasize that the time history of $C_D$ and $C_L$ is not sufficient to make such judgement, as illustrated in the low right corner plot that compares the values obtained in each period with the average value of the last 8 cycles.

4.1.2 Discretization errors

As mentioned above, each flow condition (domain size and selected boundary conditions) is calculated with 5 systematically refined grids/time steps. Therefore, the numerical uncertainty can be estimated with the procedure proposed in [33]. However, as illustrated below, the two coarsest grids/largest time steps are too coarse to obtain reliable estimations of the numerical error with power series expansions. Unavoidably [33], the estimated uncertainties including the data of these grids will be most likely too conservative. Therefore, the figures below present the lines fitted in the least squares sense to the five data points, but we will not present the estimated error bars. Nonetheless, we present the data range of the several functional flow quantities addressed in this study in table 4.1. Furthermore, table 4.1 includes the values for $1 \leq r_i \leq 2$ and for $1 \leq r_i \leq 1.5$.

The data included in table 4.1 show that the numerical uncertainty is not negligible for many of the
Figure 4.1: Illustration of the iterative convergence of the calculation of the two-dimensional flow around a squared cylinder of rounded corners at $Re = 1.74 \times 10^5$. $N_{\text{iterations}}$ is the number of iterations performed at each time step. $\phi_{\text{avg}}$ is the value of variable $\phi$ obtained from the average of the last 8 cycles. Finest grid of DM size with IM inlet conditions and pressure fixed at the top left corner of the domain.

flow conditions tested. Nevertheless, there is a systematic reduction of the data range when its values are calculated with $1 \leq r_i \leq 1.5$ instead of $1 \leq r_i \leq 2$.

It should be mentioned that roughly 5 days of c.p.u. time are required to calculate 200 dimensionless time units in the finest grid/smallest time-step using 4 quad-cores (16 processors) of 2.6GHz. A calculation performed with the same iterative convergence criteria and $r_i = 0.75$ took nearly 9 days. Unfortunately, for that level of grid/time refinement $Re_{\text{max}} < 10^{-6}$ is not sufficient to obtain a negligible level of the iterative error ($\langle C_L \rangle_{\text{avg}}$ was of the order $10^{-4}$, whereas all the present calculations exhibit $\langle C_L \rangle_{\text{avg}} \approx 10^{-6}$).

### 4.2 Domain size and pressure boundary conditions

Figure 4.2 presents the average drag coefficient $\langle C_D \rangle_{\text{avg}}$ as a function of the grid/time refinement ratio $r_i$ and the average surface pressure coefficient $\langle C_p \rangle_{\text{avg}}$ distribution for $r_i = 1$. Results were obtained
in computational domains with three different sizes (see table 3.1) and two alternative pressure boundary conditions: pressure fixed at the top left corner of the domain and pressure fixed at the external boundary.

The results show a significant influence of the domain size with the difference between the $(C_D)_{avg}$ of the smallest (DS) and largest (DL) domains reaching values close to 11%, which is significantly larger than the data range given in table 4.1. On the other hand, the results obtained with the two types of pressure boundary conditions are very similar for the three domain sizes. The $(C_p)_{avg}$ distributions show that the pressure peak at the cylinder left corners and the pressure level in the flow separation region are the main origin of the discrepancies obtained in the $(C_D)_{avg}$ predictions.

The comparison of the maximum lift coefficient $(C_L)_{max}$ and $C_p$ surface distributions at the time corresponding to $(C_L)_{max}$ for the same flow conditions of the previous figure is presented in figure 4.3. As for $(C_D)_{avg}$, there is a clear influence of the domain size on the predicted $(C_L)_{max}$. However, in this case the change is not monotonic. The justification for such behaviour of $(C_L)_{max}$ is given by the $C_p$ surface distribution that shows the same trend for the pressure on the lower part of the cylinder, whereas $C_p$ on the upper part of the cylinder exhibits a monotonic change with the domain size. On the other hand, the sharp pressure peaks at the corners of the cylinder ($\theta = \pm 45^\circ$) show that it is not trivial to obtain negligible levels of discretization errors for this flow.

Figure 4.4 presents the skin friction coefficient $C_f$ on the cylinder surface for the two conditions discussed above: average of a complete cycle and time corresponding to maximum lift coefficient $(C_L)_{max}$.
The results show a small influence of the domain size and pressure boundary conditions on the friction force at the cylinder surface. The average flow exhibits separation at the corners of the cylinder without re-attachment. Naturally, at the time corresponding to \( (C_L)_{\text{max}} \) the \( C_f \) distribution is not symmetric (the stagnation point at the back of the cylinder is located at \( y \approx -0.25D \)), but there is no re-attachment on the top and bottom surfaces of the cylinder.

For the sake of completeness, figure 4.5 presents the streamlines at the times of maximum \( (C_L)_{\text{max}} \) and zero lift coefficient \( (C_L = 0 \text{ and growing}) \) for the three selected domains for the calculations performed with pressure fixed at the top left corner of the domain. The differences between the DS and DM domains are clearly smaller than those obtained between the DM and the DL domains. It is also clear that the near-wake of these two time instants is significantly different.

The Strouhal number \( St \) and the root mean squared of the lift coefficient \( (C_L)_{\text{rms}} \) are depicted in figure 4.6 as a function of the grid/time refinement ratio \( r_i \). The change in Strouhal number with the domain size is monotonic. However, the effect of the pressure boundary conditions is mainly visible for the smallest domain DS. In fact, the change in \( St \) between domains DL and DM is smaller than the effect of the pressure boundary condition on \( St \) for the smallest domain DS. On the other hand, \( (C_L)_{\text{rms}} \) does not exhibit a monotonic change with the increase of the domain size and only the largest domain DL exhibits a negligible influence of the pressure boundary condition. Interestingly, the grid/time dependency of \( (C_L)_{\text{rms}} \) is strongly dependent on the domain size with the largest dependency obtained for the smallest values of \( (C_L)_{\text{rms}} \).
4.3 Inlet turbulence quantities

The evaluation of the influence of the level of the inlet turbulence quantities (see table 3.2) was performed for the smallest domain DS with pressure fixed at the top left corner and for the largest domain DL with pressure fixed at the external boundary. As illustrated in the previous section, there is a negligible effect of the pressure boundary conditions in the DL domain. As discussed in subsection 3.3.1, \( \nu_t \) decays much slower than \( k \) and \( \omega \) in regions of uniform flow. Therefore, we will refer to the level of \( \nu_t \) at the inlet to identify the inlet turbulence quantities in the discussion below.

The average drag coefficient \( (C_D)_{avg} \) as a function of the grid/time refinement ratio \( r_i \) and the average surface pressure coefficient \( (C_p)_{avg} \) distribution for \( r_i = 1 \) are presented in figure 4.7. The influence of the inlet value of \( \nu_t \) on \( (C_D)_{avg} \) is similar in the two domains tested. There is a monotonic decrease of \( (C_D)_{avg} \) with the growth of \( \nu_t \) with largest changes between IM and IH than between IL and IM. The \( (C_p)_{avg} \) distribution shows that these differences are originated at the corner of the cylinder with very small changes in the location of the separation point producing significantly different base pressure levels.

Figure 4.8 presents the maximum lift coefficient \( (C_L)_{max} \) as a function of the grid/time refinement ratio \( r_i \) and the \( C_p \) surface distributions at the time corresponding to \( (C_L)_{max} \). For both domains tested, there is a significant influence of the inlet \( \nu_t \) on the predicted \( (C_L)_{max} \). Nevertheless, the change with the \( \nu_t \) level is monotonic. The \( C_p \) distribution at the time of maximum lift shows that the differences are again originated at the separation points of the flow with a stronger effect of the inlet \( \nu_t \) at the top corner than at the low corner.

Figure 4.10 presents the average skin friction coefficient \( (C_f)_{avg} \) on the cylinder surface and the \( C_f \) distribution for the time of \( (C_L)_{max} \). Although there are some differences at the left corners of the
Figure 4.4: Skin friction coefficient $C_f$ on the cylinder surface: average value (left); value at time corresponding to $(C_L)_{max}$. ($\theta = 0$ at $x = -0.5D, y = 0$, $\theta = 180^\circ$ at $x = 0.5D, y = 0$, $\theta = -90^\circ$ at $x = 0, y = -0.5D$ and $C_f$ multiplied by -1 for negative $\theta$). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at $Re = 1.74 \times 10^5$ with three different sizes of the computational domain and different pressure boundary conditions. IM inlet conditions.

cylinder for both distributions, the data show that the effect of the inlet $\nu_t$ on the friction force is clearly smaller than that obtained for the pressure force.

Figure 4.11 presents the streamlines at the times of maximum $(C_L)_{max}$ and zero $(C_L = 0$ and growing) lift coefficient for the three levels of inlet $\nu_t$ obtained in the domain DS with pressure fixed at the top left corner of the domain. Although the overall pattern of the flow is similar for the three levels of inlet $\nu_t$ tested, there is a strong effect of the inlet turbulence quantities on the near wake for both time instants. Similar effects of the inlet $\nu_t$ are obtained in the DL domain with pressure fixed at the external boundary.

For the sake of completeness, figure 4.12 present the eddy-viscosity isolines for the same time instants of figure 4.11 corresponding to maximum $(C_L)_{max}$ and zero $(C_L = 0$ and growing) lift coefficient. In both time instants, $\nu_t$ is larger than $\nu$ upstream of the cylinder for the IH inlet boundary conditions. However, the most striking feature of the plots is the impact of the inlet turbulence quantities on the level of $\nu_t$ in the near-wake.

Finally, figure 4.13 presents the Strouhal number $St$ and $(C_L)_{rms}$ as a function of the grid/time refinement ratio $r_i$. The effect of the inlet $\nu_t$ on the Strouhal number is larger for the DL domain than for the DS domain where the differences between the three solution are clearly below the numerical uncertainty. On the other hand, $(C_L)_{rms}$ shows a systematic decrease with the growth of the inlet $\nu_t$ in both domains.

4.4 Overall comparison

The two previous sections demonstrate that the computational domain size, pressure boundary conditions and inlet turbulence quantities may have a significant influence on the predicted flow field around
Figure 4.5: Streamlines at times corresponding to \((C_L)_{\text{max}}\) (left) and \(C_L = 0\) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(R_n = 1.74 \times 10^5\) with three different sizes of the computational domain and pressure fixed at 1 point. IM inlet conditions.

\[
\Delta \phi(\%) \quad \text{Domain Size} \quad \text{Pressure Boundary Condition} \quad \text{Inlet Turbulence Quantities} \quad \text{Global}
\begin{array}{cccc}
(C_D)_{\text{avg}} & 9.33 & 0.10 & 5.98 & 15.2 \\
(C_L)_{\text{max}} & 11.6 & 3.47 & 31.9 & 38.3 \\
(C_L)_{\text{rms}} & 11.8 & 3.61 & 32.4 & 38.7 \\
St & 4.96 & 1.50 & 1.11 & 5.63 \\
\end{array}
\]

Table 4.2: Largest differences of average drag coefficient \((C_D)_{\text{avg}}\), maximum \((C_L)_{\text{max}}\) and root mean squared \((C_L)_{\text{rms}}\) lift coefficient and Strouhal number \(St\). Calculation of the two-dimensional flow around a squared cylinder with rounded corners at \(R_n = 1.74 \times 10^5\) with different domain sizes, pressure boundary conditions and inlet turbulence quantities. The final column corresponds to the comparison of all calculations performed.

The comparison of the data included in tables 4.1 and 4.2 shows that the discrepancies between results obtained with different problem settings (table 4.1) are clearly larger than the data range (table 4.2) and most likely larger than the numerical uncertainty. It is also evident that the domain size has the strongest influence on \((C_D)_{\text{avg}}\) and \(St\). On the other hand, the inlet turbulence quantities lead to the largest changes on \((C_L)_{\text{max}}\) and \((C_L)_{\text{rms}}\). The pressure boundary conditions have the smallest influence on the selected flow quantities.
Figure 4.6: Strouhal number $St$ (left) and root mean squared of the lift coefficient $(C_L)_{rms}$ as a function of the grid/time refinement ratio (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at $Re = 1.74 \times 10^5$ with three different sizes of the computational domain and different pressure boundary conditions. IM inlet conditions.

Figure 4.7: Average drag coefficient $(C_D)_{avg}$ as a function of the grid/time refinement ratio (left); average surface pressure coefficient $(C_p)_{avg}$ distribution ($\theta = 0$ at $x = -0.5D$, $y = 0$ and $\theta = 180^\circ$ at $x = 0.5D$, $y = 0$) for $r_i = 1$ (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at $Re = 1.74 \times 10^5$ with three different levels of the inlet turbulence quantities. Domain size and pressure boundary conditions given in the plots legend.
Figure 4.8: Maximum lift coefficient \((C_L)_{\text{max}}\) as a function of the grid/time refinement ratio (left); surface pressure coefficient \((C_p)_{\text{avg}}\) distribution \((\theta = 0\) at \(x = -0.5D, y = 0\), \(\theta = 180^\circ\) at \(x = 0.5D, y = 0\) and \(\theta = -90^\circ\) at \(x = 0, y = -0.5D)\) at the time corresponding to \((C_L)_{\text{max}}\) for \(r_i = 1\) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(Rn = 1.74 \times 10^5\) with three different levels of the inlet turbulence quantities. Domain size and pressure boundary conditions given in the plots legend.

Figure 4.9: Skin friction coefficient \(C_f\) on the cylinder surface: average value (left); value at time corresponding to \((C_L)_{\text{max}}\). \((\theta = 0\) at \(x = -0.5D, y = 0\), \(\theta = 180^\circ\) at \(x = 0.5D, y = 0\), \(\theta = -90^\circ\) at \(x = 0, y = -0.5D)\) and \(C_f\) multiplied by -1 for negative \(\theta\). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at \(Rn = 1.74 \times 10^5\) with three different levels of the inlet turbulence quantities. Domain size and pressure boundary conditions given in the plots legend.
Figure 4.10: Streamlines at times corresponding to $C_{L,\text{max}}$ (left) and $C_L = 0$ (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at $R_n = 1.74 \times 10^5$ with three different levels of the inlet turbulence quantities. Smallest domain size DS with pressure fixed at 1 point.

Figure 4.11: Eddy-viscosity ($\nu_t$) isolines at times corresponding to $C_{L,\text{max}}$ (left) and $C_L = 0$ (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at $R_n = 1.74 \times 10^5$ with three different levels of the inlet turbulence quantities. Smallest domain size DS with pressure fixed at 1 point.
Figure 4.12: Strouhal number $St$ (left) and root mean squared of the lift coefficient $(C_L)_{rms}$ as a function of the grid/time refinement ratio (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at $Re = 1.74 \times 10^5$ with three different levels of the inlet turbulence quantities. Domain size and pressure boundary conditions given in the plots legend.
Chapter 5

Results for different Reynolds numbers

5.1 Numerical uncertainty

5.1.1 Iterative errors

The two levels of $c_{it}$ mentioned above were tested for two different Reynolds number with two different sizes of the computational domain. It is important to recall that $c_{it}$ stands for the maximum normalized residual of all transport equations solved at a given time step. Figure 5.1 illustrates the typical iterative convergence results with the data obtained for the finest grid of the DT domain at $Rn = 1.74 \times 10^5$. The four top plots present the $L_2$ and $L_\infty$ norms of the final normalized residuals of all equations solved at each time step. For each convergence criteria, the $L_2$ norm of all normalized residuals is at least one order magnitude below the accepted tolerance. These four pictures also include the number of iterations ($N_{iterations}$) performed at each time, which remains between 10 and 60 iterations for $c_{it} = 10^{-3}$, and 200 and 500 iterations for $c_{it} = 10^{-6}$ when the solution becomes periodic. The two bottom plots of figure 5.1 present the lift $C_L$ and drag $C_D$ coefficients as a function of time dimensionless for calculations using different values of the iterative convergence criteria of each time step, on the other hand, demonstrate that 200 dimensionless time units are sufficient to obtain a solution with a negligible influence of the initial condition.

In order to give a quantitative idea dependence on iterative convergence criteria in the flow quantities, table 5.1 shows the values of the mean drag coefficient $(C_D)_\text{avg}$, maximum $(C_L)_\text{max}$, root mean squared $(C_L)_\text{rms}$ of the lift coefficient, root mean squared $(C_D)_\text{rms}$ of the drag coefficient, Strouhal number $St$ and base pressure coefficient ($\theta = 180^\circ$) were obtained from the average of the last eight calculated cycles.

In relation to the case shown above, it was found that there is a large influence of the iterative convergence criteria in flow quantities. The influence is mainly visible on $(C_D)_\text{rms}$ that reaches 247.7% difference between the solutions obtained with $c_{it} = 10^{-3}$ and $c_{it} = 10^{-6}$. Figure 5.2 presents the
average surface pressure coefficient \((C_p)_{avg}\) and the skin friction coefficient \((C_f)_{avg}\) distribution for \(r_i = 1\). There is a significant influence of \(c_{it}\) on the \(C_p\) level of the separated flow region, especially for the base pressure coefficient. However, the location of the separation point does not seem to be significantly affected by \(c_{it}\).

### 5.1.2 Discretization errors

For each flow condition (domain and Reynolds number) is calculated with 5 systematically refined grid/time steps. All simulations are carried out with \(c_{it} = 10^{-6}\), and the procedure used to estimate the order grid/time convergence and the numerical uncertainty is a generalized version of the procedure presented in [33]. The data range of the flow quantities considered in this study are presented in table 5.3 and 5.4 depending on the level of refinement considered, i.e. for \(1 \leq r_i \leq 1.71\) and for \(1 \leq r_i \leq 1.33\), respectively. These estimated numerical uncertainties demonstrate that the present level of grid/time refinement is too coarse to obtain negligible levels of numerical uncertainty with this model. However, we cannot go to finer refinements due to high computational costs as mentioned previously.
<table>
<thead>
<tr>
<th>$1 \leq r_i \leq 1.71$</th>
<th>Large domain</th>
<th>Tank domain</th>
</tr>
</thead>
<tbody>
<tr>
<td>Variable</td>
<td>$R_n = 7.24 \times 10^4$</td>
<td>$R_n = 9.71 \times 10^4$</td>
</tr>
<tr>
<td>$\Delta(C_D)_{avg}$ %</td>
<td>3.7</td>
<td>4.1</td>
</tr>
<tr>
<td>$\Delta(C_L)_{rms}$ %</td>
<td>13.4</td>
<td>12.4</td>
</tr>
<tr>
<td>$\Delta(C_L)_{max}$ %</td>
<td>12.9</td>
<td>12.0</td>
</tr>
<tr>
<td>$\Delta(C_D)_{rms}$ %</td>
<td>39.2</td>
<td>36.3</td>
</tr>
<tr>
<td>$\Delta St$ %</td>
<td>0.4</td>
<td>0.9</td>
</tr>
<tr>
<td>$\Delta(C_p)_{base}$ %</td>
<td>6.3</td>
<td>6.4</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Variable</th>
<th>$R_n = 2.34 \times 10^5$</th>
<th>$R_n = 2.70 \times 10^5$</th>
<th>$R_n = 3.13 \times 10^5$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Delta(C_D)_{avg}$ %</td>
<td>3.0</td>
<td>2.5</td>
<td>6.8</td>
</tr>
<tr>
<td>$\Delta(C_L)_{rms}$ %</td>
<td>2.4</td>
<td>11.1</td>
<td>93.8</td>
</tr>
<tr>
<td>$\Delta(C_L)_{max}$ %</td>
<td>2.6</td>
<td>28.3</td>
<td>101.3</td>
</tr>
<tr>
<td>$\Delta(C_D)_{rms}$ %</td>
<td>22.3</td>
<td>191.5</td>
<td>4064.7</td>
</tr>
<tr>
<td>$\Delta St$ %</td>
<td>2.6</td>
<td>2.8</td>
<td>69.0</td>
</tr>
<tr>
<td>$\Delta(C_p)_{base}$ %</td>
<td>3.6</td>
<td>2.5</td>
<td>81.7</td>
</tr>
</tbody>
</table>

Table 5.2: Data range in percentage of the selected functional flow quantities of calculation of the two-dimensional flow around a squared cylinder with rounded corners for different Reynolds numbers, $1 \leq r_i \leq 1.71$.

### 5.2 Numerical results

The numerical simulations of the flow around square cylinder with rounded corners at six Reynolds number, $R_n = \{7.24 \times 10^4, 9.71 \times 10^4, 1.74 \times 10^5, 2.34 \times 10^5, 2.70 \times 10^5, 3.13 \times 10^5\}$ have been performed for two domain sizes. However, the results were presented in detail are calculated with the corresponding size to the towing tank. For later they were compared with the published experimental results.

For each Reynolds number, the flow is computed by employing the grid/time refinement specified in table 5.4. Sensitivity tests have shown the vortex shedding results to be very sensitive to the wall discretization resolution. For this reason, it was redefined for the present study.

The time histories of the force coefficients are shown in figure 5.5 for six values of $R_n$ with finest refinement level, $r_i = 1$.

Table 5.3 summarizes the results for the flow quantities calculated for the flow around square cylinder with rounded rounded corners for the finest refinement. All these flow quantities were obtained from data of the last eight cycles performed in each calculation. It can be observed that a “drag crisis” appears for $R_n$ between $7.24 \times 10^4$ and $3.13 \times 10^5$. Across the critical regime the average drag coefficient drops...
Table 5.3: Data range in percentage of the selected functional flow quantities of calculation of the two-dimensional flow around a squared cylinder with rounded corners for different Reynolds numbers, $1 \leq r_i \leq 1.33$.

From about $1.591$ to $1.486$, the root-mean squared of the lift and drag coefficients decreases significantly up to $R_n = 2.34 \times 10^5$ and rapidly increases for higher Reynolds numbers, and the Strouhal number practically doubles jumping from $0.146$ to $0.261$.

Figure 5.4 presents the average drag coefficient $(C_D)_{avg}$ as function of the grid/time refinement ratio $r_i$ and the average surface pressure coefficient $(C_p)_{avg}$ distribution for $r_i = 1$. The results were obtained computationally for each Reynolds number with tank domain size, DT. The $(C_p)_{avg}$ distributions show that the pressure peak at the top left corners and the pressure level in the flow separation region are

<table>
<thead>
<tr>
<th>$1 \leq r_i \leq 1.33$</th>
<th>Large domain</th>
<th>Tank domain</th>
</tr>
</thead>
<tbody>
<tr>
<td>Variable</td>
<td>$R_n = 7.24 \times 10^4$</td>
<td>$R_n = 9.71 \times 10^4$</td>
</tr>
<tr>
<td>$(C_D)_{avg%}$</td>
<td>1.6</td>
<td>1.8</td>
</tr>
<tr>
<td>$(C_L)_{rms%}$</td>
<td>6.8</td>
<td>6.7</td>
</tr>
<tr>
<td>$(C_L)_{max%}$</td>
<td>6.6</td>
<td>6.6</td>
</tr>
<tr>
<td>$(C_D)_{rms%}$</td>
<td>22.6</td>
<td>21.5</td>
</tr>
<tr>
<td>$\Delta St%$</td>
<td>0.4</td>
<td>0.1</td>
</tr>
<tr>
<td>$(C_p)_{base%}$</td>
<td>3.1</td>
<td>3.1</td>
</tr>
<tr>
<td>Variable</td>
<td>$R_n = 2.34 \times 10^5$</td>
<td>$R_n = 2.70 \times 10^5$</td>
</tr>
<tr>
<td>$(C_D)_{avg%}$</td>
<td>1.5</td>
<td>1.0</td>
</tr>
<tr>
<td>$(C_L)_{rms%}$</td>
<td>0.1</td>
<td>5.5</td>
</tr>
<tr>
<td>$(C_L)_{max%}$</td>
<td>0.4</td>
<td>5.4</td>
</tr>
<tr>
<td>$(C_D)_{rms%}$</td>
<td>11.2</td>
<td>76.1</td>
</tr>
<tr>
<td>$\Delta St%$</td>
<td>1.3</td>
<td>1.4</td>
</tr>
<tr>
<td>$(C_p)_{base%}$</td>
<td>1.8</td>
<td>0.9</td>
</tr>
</tbody>
</table>

Table 5.4: Parameters of the refinement level used to calculate flow around squared cylinder of rounded corners for all Reynolds numbers.
Table 5.5: Flow quantities of the refinement level used to calculate flow around squared cylinder of rounded corners for all Reynolds numbers.

<table>
<thead>
<tr>
<th>( Rn )</th>
<th>((C_D)_{avg})</th>
<th>((C_L)_{rms})</th>
<th>((C_L)_{max})</th>
<th>((C_L)_{rms})</th>
<th>( St )</th>
<th>((C_P)_{base})</th>
</tr>
</thead>
<tbody>
<tr>
<td>(7.24 \times 10^4)</td>
<td>1.591</td>
<td>0.938</td>
<td>1.310</td>
<td>0.069</td>
<td>0.146</td>
<td>-1.224</td>
</tr>
<tr>
<td>(9.71 \times 10^4)</td>
<td>1.581</td>
<td>0.881</td>
<td>1.233</td>
<td>0.054</td>
<td>0.147</td>
<td>-1.207</td>
</tr>
<tr>
<td>(1.74 \times 10^5)</td>
<td>1.544</td>
<td>0.720</td>
<td>1.013</td>
<td>0.023</td>
<td>0.149</td>
<td>-1.158</td>
</tr>
<tr>
<td>(2.34 \times 10^5)</td>
<td>1.492</td>
<td>0.560</td>
<td>0.792</td>
<td>0.008</td>
<td>0.150</td>
<td>-1.103</td>
</tr>
<tr>
<td>(2.70 \times 10^5)</td>
<td>1.486</td>
<td>0.673</td>
<td>0.984</td>
<td>0.116</td>
<td>0.261</td>
<td>-1.725</td>
</tr>
<tr>
<td>(3.13 \times 10^5)</td>
<td>1.488</td>
<td>0.754</td>
<td>1.127</td>
<td>0.142</td>
<td>0.256</td>
<td>-1.834</td>
</tr>
</tbody>
</table>

the main origin of the discrepancies obtain in the \((C_D)_{avg}\) predictions. However, these differences for the \((C_p)_{avg}\) distributions are originated at the re-attachment on the back surface of the cylinder or if the re-attachment on the top and bottom surfaces of the cylinder. The negative average pressures on side surfaces increase with the Reynolds number, even if the flow does not re-attach to the side surfaces.

Note that, for two low Reynolds number was not considered some refinement levels for the estimation of the numerical uncertainty, because the flows around a squared cylinder of rounded corners that were performed for these refinement levels are different from calculated for the finest refinement level, i.e. the flow regime is not symmetric along the vortex shedding period and that was why they were not showed. At the procedure used to estimate numerical uncertainty to the flow around the square cylinder with rounded corners for the \( Rn = 2.70 \times 10^5\), it was only considered the results obtained for the last three refinement level, since the solutions to the flow regime were not the same for others refinement levels.

The comparison of the maximum lift coefficient \((C_L)_{max}\) and \( C_p \) surface distributions at the corresponding to \((C_L)_{max}\) for the same flow conditions of the previous figure 5.5. The \( C_p \) distribution at the time of maximum lift shows that the differences are again originated at the separation points of the flow with a stronger effect of the Reynolds number.

Figure 5.6 presents the average skin friction \((C_f)_{avg}\) on the cylinder surface and the \( C_f \) distribution for the time of \((C_L)_{max}\). Meanwhile, with increasing the Reynolds number, the average separated flow can re-attach to the side surface with ease.

Figure 5.7 presents the streamlines at the times of maximum \(((C_L)_{max})\) and zero \((C_L = 0\) and growing) lift coefficient for two Reynolds number obtained in the domain DT with pressure fixed at the top left corner of the domain.

Finally, figure 5.8 presents the Strouhal number \( St \) and \((C_L)_{rms}\) as a function of the grid/time refinement ratio \( r_i \). The effect of the Reynolds number on the Strouhal number is larger for the flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT.

Figure 5.9 demonstrates that the transition between subcritical and supercritical regime appear to be very sensitive to the test conditions, namely for this case the domain size.

### 5.3 Experimental data

Recently at MARIN, several experimental campaigns have been performed to access the VIV (vortex induced vibrations) phenomenon for fixed and freely vibration, smooth square cylinder with rounded
corners, the first measurements being made [9]. These experiments were done with the High Reynolds VIV test apparatus in the High Speed Basin on a 200mm rounded-off square cylinder with 18mm corner radius and provided time traces of the forces on the square cylinder, \((C_D)_{avg}\) and \(St\). With respect to the lift coefficient \(C_L\), maximum value is difficult to be defined the irregular behavior of the measured signals. This is caused by the three-dimensional effects observed in the basin. Table 5.6 summarizes the experimental results for the flow quantities, \((C_D)_{avg}\) and \(St\).

<table>
<thead>
<tr>
<th>(R_e)</th>
<th>((C_D)_{avg})</th>
<th>(St)</th>
<th>((C_D)_{avg})</th>
<th>(St)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(7.24 \times 10^4)</td>
<td>1.579681</td>
<td>0.135179</td>
<td>1.634260</td>
<td>0.133835</td>
</tr>
<tr>
<td>(9.71 \times 10^4)</td>
<td>1.583371</td>
<td>0.133005</td>
<td>1.645611</td>
<td>0.131777</td>
</tr>
<tr>
<td>(1.74 \times 10^5)</td>
<td>1.689359</td>
<td>0.135462</td>
<td>1.694741</td>
<td>0.136630</td>
</tr>
<tr>
<td>(2.34 \times 10^5)</td>
<td>1.347625</td>
<td>0.118064</td>
<td>1.361771</td>
<td>0.123867</td>
</tr>
<tr>
<td>(2.70 \times 10^5)</td>
<td>1.363977</td>
<td>0.116691</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>(3.13 \times 10^5)</td>
<td>1.587285</td>
<td>0.128321</td>
<td>1.608560</td>
<td>0.131954</td>
</tr>
</tbody>
</table>

Table 5.6: Experimental data for the flow quantities, \((C_D)_{avg}\) and \(St\).

5.3.1 Experimental uncertainty

In order to make a consistent validation procedure, we must also determine the uncertainties in the experimental data. However, it is quite rare to see experimental uncertainties in publications, thus one attempt to obtain some estimation of these uncertainties from the experimental data. The estimation more reliable check of the statistical convergence of the mean value of the selected flow quantities may be performed with the auto-covariance method proposed [40] in that estimates the uncertainty of the mean value of a time series as a function of experience time.

5.4 Validation exercise

A validation exercise was also done for the calculations showed herein in order to evaluate our approach. According to [41], the aim of Validation is to estimate the modelling error of the mathematical model to set of experimental data, which represents the physical model. Once validation is done, one can say that the model/code is valid for that particular problem and conditions. The procedure proposed by ASME [34] for validation is based on the comparison of the quantities:

\[ U_{val} = \sqrt{U_{num}^2 + U_{input}^2 + U_D^2}, \]  

(5.1)

and

\[ E = S - D. \]  

(5.2)

In Eq. (5.1) and Eq. (5.2), \(U_{num}\) is the numerical uncertainty estimated for a certain quantity, \(U_{input}\) is the uncertainty of the parameter inputs (which are considered negligible for the present exercise) and \(U_D\) is the experimental uncertainty, \(S\) is the numerical prediction of the parameter value and \(D\) is the
Table 5.7: Validation results for drag coefficient, $(C_D)_{avg}$.

<table>
<thead>
<tr>
<th>$Rn$</th>
<th>$U_{num}$ (%)</th>
<th>$U_D$ (%)</th>
<th>$U_{val}$</th>
<th>$E$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$7.24 \times 10^4$</td>
<td>0.113 (7.08)</td>
<td>0.011 (0.72)</td>
<td>0.113</td>
<td>0.011</td>
</tr>
<tr>
<td>$9.71 \times 10^4$</td>
<td>0.073 (4.63)</td>
<td>0.007 (0.43)</td>
<td>0.074</td>
<td>0.002</td>
</tr>
<tr>
<td>$1.74 \times 10^5$</td>
<td>0.287 (18.57)</td>
<td>0.007 (0.43)</td>
<td>0.287</td>
<td>0.146</td>
</tr>
<tr>
<td>$2.34 \times 10^5$</td>
<td>0.049 (3.31)</td>
<td>0.007 (0.50)</td>
<td>0.050</td>
<td>0.145</td>
</tr>
<tr>
<td>$2.70 \times 10^5$</td>
<td>0.007 (0.45)</td>
<td>0.006 (0.40)</td>
<td>0.009</td>
<td>0.122</td>
</tr>
<tr>
<td>$3.13 \times 10^5$</td>
<td>0.103 (6.90)</td>
<td>0.017 (1.07)</td>
<td>0.104</td>
<td>0.099</td>
</tr>
</tbody>
</table>

The comparison between $U_{val}$ and $E$ may lead to two possibilities:

- $|E| \gg U_{val}$ means that the comparison is poor most likely because the modelling errors are of most importance.
- $|E| < U_{val}$ means that the solution is within the noise imposed by the different sources of uncertainty. In this case, if $E$ is small enough, then the solution is validated with the experiment at $U_{val}$ precision. Otherwise, the quality of the numerical solution and/or the experiment should be improved for a better comparison.

Table 5.7 shows the results for our validation exercise concerning drag coefficient, considering the forward towing tests for estimates the experimental uncertainties.
Figure 5.1: Illustration of the iterative convergence of the calculation of the two-dimensional flow around a squared cylinder of rounded corners at $Re = 1.74 \times 10^5$. $N_{\text{iterations}}$ is the number of iterations performed at each time step. Time history of the force coefficients, as a function of iterative converge criteria, $c_{it}$: $10^{-3}$ (left) and $10^{-6}$ (right). Finest grid of DT size domain.
Figure 5.2: Average surface pressure coefficient $(C_p)_{avg}$ (left) and average skin friction coefficient $(C_f)_{avg}$ (right) distribution ($\theta = 0$ at $x = -0.5D, y = 0$ and $\theta = 180^\circ$ at $x = 0.5D, y = 0$) for $r_i = 1$. Calculation of the two-dimensional flow around a squared cylinder of rounded corners at $R\text{e} = 1.74 \times 10^5$ with DT size of the computational domain. Solid lines for $c_{it} = 10^{-6}$ and dashed lines for $c_{it} = 10^{-3}$.
Figure 5.3: Time histories of the drag coefficient (red line) and the lift coefficient (blue line) for six values of $Rn$ with finest refinement level, $r_i = 1$. 
Figure 5.4: Average drag coefficient \( (C_D)_{\text{avg}} \) as a function of the grid/time refinement ratio (left); average surface pressure coefficient \( (C_p)_{\text{avg}} \) distribution \( \theta = 0 \) at \( x = -0.5D, y = 0 \) and \( \theta = 180^\circ \) at \( x = 0.5D, y = 0 \) for \( r_i = 1 \) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT. Reynolds number given in the plots legend.

Figure 5.5: Maximum lift coefficient \( (C_L)_{\text{max}} \) as a function of the grid/time refinement ratio (left); surface pressure coefficient \( (C_p)_{\text{avg}} \) distribution \( \theta = 0 \) at \( x = -0.5D, y = 0 \), \( \theta = 180^\circ \) at \( x = 0.5D, y = 0 \) and \( \theta = -90^\circ \) at \( x = 0, y = -0.5D \) at the time corresponding to \( (C_L)_{\text{max}} \) for \( r_i = 1 \) (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT. Reynolds number given in the plots legend.
Figure 5.6: Skin friction coefficient $C_f$ on the cylinder surface: average value (left); value at time corresponding to $(C_L)_{max}$, $(\theta = 0$ at $x = -0.5D, y = 0$, $\theta = 180^\circ$ at $x = 0.5D, y = 0$, $\theta = -90^\circ$ at $x = 0, y = -0.5D$ and $C_f$ multiplied by -1 for negative $\theta$). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT. Reynolds number given in the plots legend.
\[ Rn = 1.74 \times 10^5 \ (C_L)_{max} \]

\[ Rn = 2.70 \times 10^5 \ (C_L)_{max} \]

Figure 5.7: Streamlines at times corresponding to \((C_L)_{max}\) (top) and \(C_L = 0\) (bottom). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT. Reynolds number given in the plots legend.
Figure 5.8: Strouhal number $St$ (left) and root mean squared of the lift coefficient $(C_L)_{rms}$ as a function of the grid/time refinement ratio (right). Calculation of the two-dimensional flow around a squared cylinder of rounded corners at various Reynolds numbers for tank domain size, DT. Reynolds number given in the plots legend.

Figure 5.9: Influence the domain size at the transition between subcritical and supercritical regime for the flow quantities.
Chapter 6

Conclusions

This thesis presents an investigation of the flow around a squared cylinder with rounded corners based on numerical simulations. The flow is predicted with one of the simplest mathematical models available: two-dimensional, incompressible, Reynolds-averaged Navier-Stokes equations supplemented with the SST $k − \omega$ eddy-viscosity model. Naturally, ensemble average has been applied due to the periodic nature of the flow.

Two different topics are addressed in this work:

- The effect of the problem settings on the predicted flow quantities, i.e. the domain size and the specification of the pressure and inlet turbulence quantities boundary conditions.

- The effect of the Reynolds number for the range $7.24 \times 10^4$ to $3.13 \times 10^5$ on the flow regimes.

The main quantities of interest selected for this study are the average drag coefficient, the maximum and root mean squared lift coefficient and the Strouhal number. Besides these functional quantities, the surface distributions of the pressure and skin friction coefficients are also analyzed to understand the origin of the differences obtained in the force coefficients.

The numerical uncertainty of the prediction has been carefully addressed including the assessment of the iterative and discretization errors contributions to the numerical uncertainty. In order to obtain a negligible contribution of the iterative error, the convergence criteria adopted for each time step must be significantly more demanding than the traditional three orders of magnitude of residual drop. For the present level of grid refinement, the iterative convergence criteria requires that the maximum normalized residual of all transport equations solved (including turbulence quantities) must be smaller than $10^{-6}$. The normalization makes the residuals of the equations equivalent to the change in the solution field for a simple Jacobi iteration. Furthermore, the time simulated for each flow condition guarantees that the influence of the initial condition on the last six cycles is negligible.

Naturally, such tight iterative convergence criteria makes grid/refinement studies to address discretization errors extremely time consuming. Nevertheless, 4 or 5 geometrically similar grids with simultaneous time refinement have been generated to allow the estimation of the numerical uncertainty based on power series expansions. Unfortunately, most of the selected flow quantities do not exhibit
monotonic convergence with grid/time refinement, which makes the estimation of the numerical uncertainty troublesome and most likely too conservative. However, it is clear from the data changes obtained in the three finest grids that the present level of grid/time refinement (1440 cells on the surface of the cylinder) leads to significant discretization errors for the selected flow quantities. The sharp gradients of the pressure and skin friction coefficients observed at the flow separation points (corners of the cylinder) suggest that extremely dense grids to obtain negligible numerical uncertainties.

The results obtained for the investigation of the problem settings suggest the following conclusions:

- The specification of the pressure boundary conditions exhibits the smallest influence on the selected flow conditions. However, for the smallest domain sizes tested, there is a clear difference between imposing pressure at the external boundaries or at a single point.

- The domain size and the specification of the inlet turbulence quantities have a significant influence on the outcome of the simulations. Interestingly, the effect of these choices is not identical for all the selected flow quantities:
  - The strongest influence of the domain size is obtained for the average drag coefficient and Strouhal number.
  - The specification of the inlet turbulence quantities has the largest effect on the maximum and root mean squared lift coefficient.

The main conclusion of this sensitivity study is that proper Validation cannot be performed without detailed information of all the problem settings including domain size and all boundary conditions.

The second exercise presented in this thesis demonstrates that the present mathematical model is able to identify two different regimes for the flow around this squared cylinder with rounded corners:

1. Flow separation occurring at the left corners of the cylinder without reattachment on the top and bottom surfaces.

2. Separation bubble on the top and bottom surfaces of the cylinder followed by flow separation at the right corners of the cylinder to generate vortex shedding.

These two regimes are dependent on the Reynolds number and its determination is only possible for a computational domain that matches the size of the towing tank where the experiments were performed. The results obtained for the second regime suggest that this type of flows may become extremely complex making the determination of the mean flow solution a real challenge. Nonetheless, it is clear that Validation of this type of flows depends on very challenging experiments that should obtain a complete definition of boundary conditions and a careful assessment of experimental uncertainties.

Naturally, future work on this topic will require the use of three-dimensional domains and more sophisticated mathematical models. However, it is evident that such predictions will be extremely time consuming and out of the scope of practical calculations. Therefore, it may be worthwhile to pursue more modest goals like reducing the discretization error to negligible levels and investigate the role of transition from laminar to turbulent flow.
References


