

Turbulent Flow Inside Longitudinally Corrugated Pipes

by

Zibo Jane

Reg No.14100480

Department of Mathematics And Statistical Sciences

Botswana International University Of Science And Technology

Email: zibo.jane@studentmail.biust.ac.bw

Tel: (+267) 72285720

A Thesis submitted to the College of Sciences in Partial Fulfilment of the
Requirements for the award of the Degree of Master of Science in Applied
Mathematics of BIUST

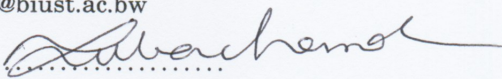
Supervisor

Dr. N. Subaschandar

Department of Mathematics and Statistical Sciences

College of Sciences, BIUST

Email: raos@biust.ac.bw

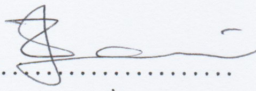
Signature: 

Date: 05/DEC/2018

December 2018

Declaration And Copyright

I, Zibo Jane, declare that this thesis titled: TURBULENT FLOW INSIDE LONGITUDINALLY CORRUGATED PIPES is my own original work and that it has not been presented and will not be presented to any other university for a similar or any other degree award.

Signature: 

Date: 05/12/2018

This thesis is copyright material protected under the Berne Convention, the Copyright Act of 1999 and other international and national enactments, in that behalf, on intellectual property. It must not be reproduced by any means, in full or in part, except for short extracts in fair dealing; for researcher private study, critical scholarly review or discourse with an acknowledgement, without the written permission of the office of the Provost, on behalf of both the author and the BIUST.

Certification

The undersigned certifies that he has read and hereby recommends for acceptance by the College of Sciences a thesis titled: TURBULENT FLOW INSIDE LONGITUDINALLY CORRUGATED PIPES, in fulfillment of the requirements for the Degree of Master of Science in Applied Mathematics of BIUST.

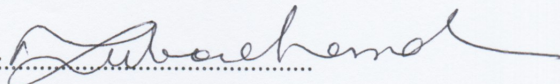
Supervisor:

Dr. N. Subaschandar

Department of Mathematics and Statistical Sciences

College of Sciences, BIUST

Email: raos@biust.ac.bw

Signature:.....

Date:.....05 DEC 2018.....

Dedication

I dedicate this work to my son Zibo Ty Jane and my lovely wife Nametso Jane.

Acknowledgements

I would like to express my deep gratitude to my supervisor Dr N. Subaschandar for fruitful discussions and valuable remarks. It is a great pleasure to acknowledge the support and encouragement I received from my wife Mrs Nametso Jane during preparation of this report. Also I want to acknowledge the support I got from my friends Dr Stanley Sekwena, Mr Tsholofelo Bafana and Mr Taelo Storey Modimoeng. Special thanks are due to God who is my strength in everything.

Contents

List of Figures	vii
List of Tables	viii
Abstract	ix
1 Introduction	1
1.1 Introduction	1
1.2 Goals of the Investigation	6
1.3 Thesis Outline	6
2 Geometry	8
3 Governing Equations	11
3.1 Basic Model and Assumptions	12
3.1.1 Assumptions	12
3.1.2 Governing Equations	12
3.2 Turbulence Models	13
3.2.1 $k - \epsilon$ Turbulence Model	16
3.2.2 $k - \omega$ Turbulence Model	18
4 Numerical Simulation	21
4.1 Computational Fluid Dynamics	21
4.1.1 CFD Background	21
4.1.2 Pre-Processor	22
4.1.3 Solver (FLUENT Solver)	23
4.1.4 Post-Processor	23

4.2	The Finite Volume Method	24
4.2.1	Face integrals	25
4.2.2	Gradient Reconstruction	26
4.3	Time Discretization	27
4.4	Boundary Conditions	28
4.4.1	Inlet	28
4.4.2	Outlet	29
4.4.3	Walls and Symmetries	29
5	Results and Discussions	33
5.1	Conservation of Mass	33
5.2	Grid Independence	33
5.3	Smooth Wall Validation	35
5.4	Results for Corrugated Wall	36
5.4.1	Flow Pattern	36
5.4.2	Friction Factor	51
5.5	Performance of Turbulence Models	54
6	Conclusion	56
6.1	Conclusion	56
6.2	Recommendations For Further Studies	57
	Bibliography	58

List of Figures

2.1	A simplified diagram of a part of geometry under study (Total length is 7λ)	9
4.1	Mesh distribution (a) in the full pipe (b) zoomed to 3λ and (c) inside the corrugation cavity	32
5.1	Computed friction factors for Smooth pipe	36
5.2	Total Pressure Contour at Re 10000	37
5.3	Total pressure contour at Re 15000	38
5.4	Total pressure contour at Re 20000	39
5.5	Velocity Contour at 10000	41
5.6	Velocity Contour at 15000	42
5.7	Velocity Contour at 20000	43
5.8	Velocity Vectors at Re 10000	44
5.9	Velocity Vectors at Re 15000	45
5.10	Velocity Vectors at Re 20000	46
5.11	Velocity vectors inside the corrugation at Re 10000	48
5.12	Velocity vectors inside the corrugation at Re 15000	49
5.13	Velocity vectors inside the corrugation at Re 20000	50
5.14	Numerical friction factor acquired for every geometric configurations and Reynolds numbers projected with smooth pipe.	51
5.15	Computed friction factors for Smooth pipe	53
5.16	Computed friction factors for Smooth pipe	53
5.17	Computed friction factor for corrugated pipe using $k - \epsilon$ and $k - \omega$ turbulence models	54

List of Tables

5.1	Grid sizes for each geometry	34
5.2	Grid independence study for the geometry $\lambda/\alpha = 1$ at Re 20 000 using $k - \epsilon$ model	34
5.3	Friction factor with $k - \epsilon$ and $k - \omega$ turbulence models	55

Abstract

Longitudinally corrugated pipes can be described as pipes with periodically converging-diverging cross-area arranged along the flow course. These pipes are mainly utilized in engineering applications, for example, pipe flow in a heat exchanger where wall corrugation improves heat convection, gas to liquid contractors in the chemical business, ventilation, air conditioning, drag reduction and mass transfer efficiency. Both experimental and analytical works detailed the impact of corrugation amplitude and distance in the turbulent flow by investigation of a few turbulent characteristics, for example, Reynolds stress and velocity variation. In our examination, we are concerned in assessing friction factor for turbulent flow inside longitudinally corrugated pipes as a function of the amplitude of the pipe. Mass and momentum conservation equations are reviewed and particular boundary conditions are fixed to describe a completely developed periodic system in a solitary axisymmetric bidimensional module which signifies the periodically corrugated pipe configuration. Two two-equation turbulence models ($k - \epsilon$ and $k - \omega$) are utilized in the calculations. The numerical simulation of the isothermal, single phase and incompressible flow is prepared utilizing Computational Fluid Dynamics methods, using the commercially available CFD software ANSYS FLUENT. To begin with, the completely developed turbulent flow in a smooth pipe is studied. Simulations are carried out to authenticate the preferred models, computational meshes and boundary conditions. Next a longitudinally corrugated pipe of length 7λ is studied. Impact of surface corrugation, controlled by wavelength-amplitude proportion λ/a on the flow is exhibited. The simulation work has been completed for $\lambda/a = 1, 11$ and 19 for a Reynolds number range of $10\,000$ to $20\,000$. Flow phenomena like flow separation and vortex caused by the corrugation are described. The friction factor for various values of the wavelength is calculated by operating a set of simulations. The friction factor for a given Reynolds number in a longitudinally corrugated pipe decreases with increase of the wavelength.

Chapter 1

Introduction

1.1 Introduction

A longitudinally corrugated pipe can be described as a pipe with periodically converging-diverging cross-area arranged along the flow course. These pipes have been studied for a long time, fundamentally in light of their substantial use in numerous engineering applications, for example, pipe flow in a heat exchanger where wall corrugation improves heat transfer, gas to fluid contractors in the chemical business, ventilation, air conditioning, drag reduction and mass transfer efficiency. In the flow entries with a periodically changing cross-area, the fluid flow achieves a completely developed system that varies mainly from that for a regular smooth flow course [1]. In the occasionally changing cross-areas, the completely matured velocity field replicates itself at the corresponding axial areas in progressive phases. The flow within corrugated pipes has some vital characteristics, which are not characteristics to flows in regular pipes. The likely impacts of corrugation is that the change from laminar to turbulent flow will happen at lower Reynolds numbers (say, at $Re < 2\,000$), than in a smooth straight pipe [2].

The investigation of flow in irregular pipes goes back to Nikuradse's experiments [3], whose outcomes obtained from artificially corrugated pipes, were afterwards organized in the more notable type of the Moody Diagram [4], where for laminar flow, the friction factor generally appeared as free of the corrugation.

In spite of this, few experimental investigations have demonstrated that the contribution of the wall roughness is not minor, even in the laminar case [5–7]. This occurs particularly when handling a wall-shape, instead of just with some random roughness. Perry et al. [8] held one of the principal hypothetical investigations about turbulent flow in distinct rough walls. Later they then suggested a geometric denomination for the distinct corrugated components founded on the corrugation length scale in relation to the flow.

Nishiruma et al. [9], Russ and Beer [10], and Yang [11] explored the transitional flow attributes in grooved channels. They described the laminar-turbulent transition to happen at low Reynolds numbers in contrast with convectional channels. Other essential feature of flow in corrugated pipes is the potential existence of local adverse pressure gradients and, subsequently, the presence of flow separations. Flow through corrugated channel works as a straightforward case of separated flow. Skin friction (friction at the wall) and the structure of the wall (drag) causes pressure loss along the pipe. In steady flow, the skin friction corresponds to the pressure loss per unit distance [12]. Said et al. [13] investigated laminar flow through a sinusoidal hole. They studied how the increase of flow velocity generates vortex within the corrugation and how it affects thermal and hydrodynamic performance. Wang and Vanka [14] detailed higher friction factor values for sinusoidal channel contrasted with the parallel plate channel of the same inside spacing.

Russ and Beer [10, 15] measured mean friction coefficient of axisymmetric pipe for three unique amplitudes and recorded frictional loss increments with the increase of amplitude at same Reynolds number. Sparrow and Hossfeld [16] performed experimental study to decide the pressure drop, heat transfer and flow field reactions to the curving of the crest of a corrugated wall pipe at a Reynolds number range from 2 000 to 33 000. They detailed that because of curving, friction factor diminishes equivalent to a given Reynolds number considerably beyond did the Nusselt number. Mahmud et al. [1] examined heat transfer, pressure drop and flow field properties within a sinusoidally changing curly pipe and they revealed that the high the surface waviness the early the flow detachment, greater the pressure drop and higher the heat transfer rate.

Yoon et al. [17] explored the impact of wave amplitude on turbulent flow in a wavy ducts utilizing Direct Numerical Simulation (DNS) with the immersed boundary method. They noticed that the pressure drag coefficient gets larger as the proportion λ/a grows, while the friction drag coefficient is maximal at the defined amplitude to wavelength ratio, $\lambda/a = 0.3$. Maass and Schumann [18] carried out DNS of turbulent flows on curvy wall utilizing the finite difference method and demonstrated the effectual friction velocity is around 50% bigger at the wavy area compare to the smooth area as a result of the extra pressure drag.

Hudson et al. [19] estimated the spatial and transient variety of the velocity elements in a flow with wavy surface utilizing Laser Doppler Velocimetry (LDV) and announced mean velocity fields and varying measures like kinetic-energy production and turbulent stresses. In their investigation, the Reynolds number in view of bulk velocity and duct height is 6 760 and the amplitude to wavelength proportion is equivalent to 0.05. The analyses demonstrated that the shear layer in the proximity of the detachment area is the region of huge Reynolds stresses and turbulent kinetic energy production. Cherukat et al. [20] led Direct Numerical Simulation investigations of the turbulent flow past a sinusoidal curly surface utilizing spectral components of seventh order. Their conclusions were approved of Hudson et al. [19] experimental data and are viewed a solid reference point for numerical simulation of wavy walls. They announced turbulent statistics and instantaneous flow fields and utilized them to discover velocity explosions in the detached-flow area that expand over huge distances away from the wall.

Nishimura et al. [9] explored flow characteristics like pressure drop, wall shear stress and flow pattern in a duct with symmetric sinusoidal wavy wall. This investigation revealed that for Reynolds number beyond 700, turbulent flow takes place, attributing to the beginning of an unstable vortex movement and friction factor is indirectly proportional to Reynolds number at the laminar range. Perry et al. [8] indicate areas with square coarse pits as “*d*-type” irregular surfaces which has the form of size of the pipe diameter, d , in inner flows. Pits or cavities with substantial lengths in regard to the pit height can’t support

steady vortices, and the external flow reattach in the base of the pit; in this arrangement, the external flow is then open to the ridges, and the boundary layer thickness is not any more a suitable roughness length scale. Presently, the pit height, k , is certainly the suitable roughness length scale because it has a considerably more critical part in outlining the wall roughness. Perry et al. [8] categorize cavities with expansive lengths in regard to the ridge height as “ k -type” roughness, and these two classifications have been extensively utilized from that point forward.

Chen et al. [21] performed an experimental investigation of turbulent air flow in ducts with corrugated surfaces. The principal focal point was to explore the feasibility of drag reduction in turbulent flows at low Reynolds numbers related with the capability of different corrugation formations, for example, trapezoidal, triangular and “ d -type” square shapes in maintain steady vortices and preventing propagation of turbulent explosions all over the flow. Actually, Chen et al. [21] develops of past findings (Liu et al. [22], Kennedy et al. [23] and Walsh & Weinstein [24]) where it was suggested that drag reduction is attainable if the limitless height, h^+ , and pitch, S^+ , of the cavities were retained below 25 and 100, accordingly, with the h^+ and S^+ stated as $h^+ = hV_*/\nu$ and $S^+ = hV_*/\nu$, in which h and S are the height and pitch of the cavities respectively, ν is the cinematic viscosity and V_* is the friction velocity. Chen et al. [21] established that for Reynolds numbers below 16 000 trapezoidal and triangular corrugation shapes with $h^+ \simeq 20$ and $S^+ \simeq 83$ are able of lowering the global friction factor, compared with that computed for a duct with smooth surfaces. For $h^+ \simeq 32$ and as yet retaining $S^+ < 100$, the friction factor inclined to be more than that computed for smooth wall ducts.

Utilising Laser-Doppler Anemometry, Djenidi et al. [25] considered the flow past a flat plate with “ d -type” coarse. The form of the outer flow past a flat plate was given by Djenidi et al. [25] where an exceptionally outlined formation which granted the visualisation of the vortices behaviour within the “ d -type” corrugation. The fundamental visible finding of Djenidi et al. [25] was that, facing Perry et al. [8], “ d -type” grooves do not completely cut off the external flow from the roughness effects. In the contrary, there are powerful ejections introduced

in the cavities enclosed vortex discharged just to the external flow, which occurs at a time scale adjacent to $5.5\delta/U_\infty$, in which U_∞ is the bulk velocity.

Such ejections symbolize a typical improvement of momentum transfer from the cavity to the external flow, and as per Djenidi et al. [25] this impact bring about global increments in turbulent forces close to the wall, comprising other entities, specifically, the friction coefficient, the Reynolds shear stress and consequently the average pressure drop near the corrugated wall. Silberman [26] projected that it is attainable to lower drag in curly corrugated pipes (comprising of cavities with a length of identical order of size as of the ridges height) by reducing the helix angle (for example, arranging ridges more attached in the flow coarse), and subsequently Silberman [27] showed his past projected inclination as being right. Then again, Dong et al. [28] demonstrated that spirally corrugated pipes with cavities significantly larger than the ridge height is likely to raise the global friction factor in contrast with smooth pipes.

Morales et al. [29, 30] directed an experimental investigation in turbulent flow in “*d*-type” corrugated pipes, with both single-start spiral and straight grooves. The experimental loop built at the LACIT Thermal Science Laboratory of UTFPR Federal Technological University Paraná (Morales et al. [29]) was intended to give turbulent water flow in 2.54 cm diameter pipes, and the acquired information demonstrated that the friction factor increases in corrugated pipes compared to smooth pipes.

A review of the literature shows that the friction factor is increased in corrugated pipes in comparison with that of smooth pipes. The review shows that flow separation occurs as flow moves downstream into the deviating part and reattaches on the approaching part of the corrugation cavity. This forms a vortex inside the corrugation cavity. The vortex starts more stable and becomes unstable as it moves downstream with the increase in Reynolds number and corrugation height in the cavity.

However, when reviewing the literature we did not find studies which clearly show what has the greatest influence on the instantaneous velocity fields (like the vortex formed in the corrugation cavity) between the Reynolds number and

the surface waviness. Hence, this present study has the goals of investigation as presented below.

1.2 Goals of the Investigation

In this investigation, the turbulent flow through longitudinally corrugated pipes with different wavelength will be studied using numerical fluid mechanics methods. In this study, we are concerned in calculating friction factor for turbulent flow inside longitudinally corrugated pipes as a function of the wavelength of the pipe. The effect of Reynolds number and surface waviness will be studied for a given wavelength and amplitude of the pipe. Additionally, instantaneous velocity fields near the wavy walls for each wavelength and amplitude are studied and their coherent forms described. The performance of two distinct two-equation turbulence models are investigated.

1.3 Thesis Outline

This report consist of six chapters.

In Chapter 2 all the three geometries investigated in this report are described. The details about the dimensions of each figure are given. Chapter 3 presents the governing equations and boundary conditions for all investigations presented. In this chapter, the Navier-Stokes equations for incompressible flow and the Reynolds averaged equations are determined. Finally, two turbulence transport models, thus the $k - \epsilon$ and $k - \omega$ turbulence models are examined and discussed. Chapter 4 outlines the basics of Computational Fluid Dynamics (CFD) method. The history of CFD and the Finite Volume Method, the solver parameters for the commercial CFD ANSYS FLUENT software and the post-processing methods are described. The time discretization in FLUENT which is achieved through a fractional step scheme is explained. The boundary conditions are prescribed for the resolution of discretized equations described previously. In Chapter 5, the computational results of the current study are presented and discussed. The simulation results are utilized to calculate the friction factor

for various wavelength of the pipe. Instantaneous velocity fields near the corrugated walls for each wavelength and their coherent structures are described. Chapter 6 presents important conclusions of the present study.

Chapter 2

Geometry

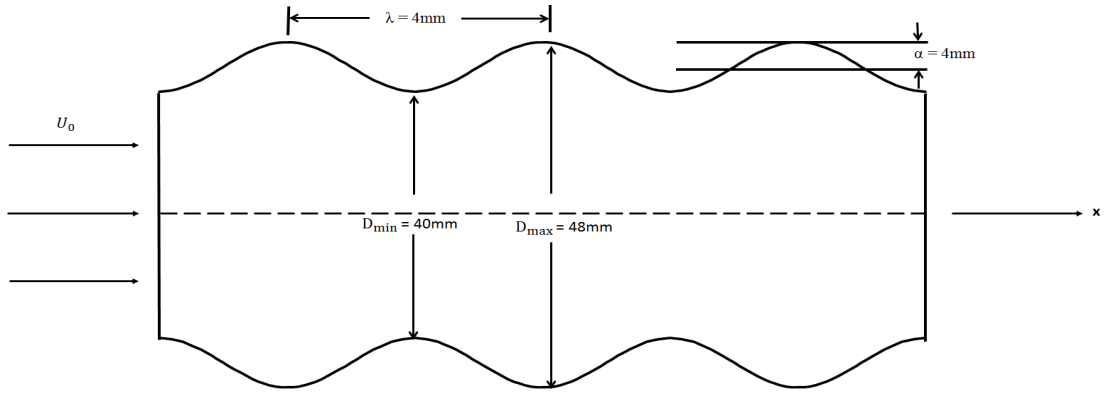
In this chapter, various geometries considered for simulation in this present investigation are presented.

A pipe of length 7λ where λ is the wavelength is considered for investigation. Figure 2.1 shows the schematic diagram of a portion of geometry used in this investigation. For simplicity, only three cycles, 3λ , are shown in Figure 2.1. The wavelength λ varies across the three pipes with amplitude a constant. The following equation defines the radius of the pipe considered, which display a sine curve.

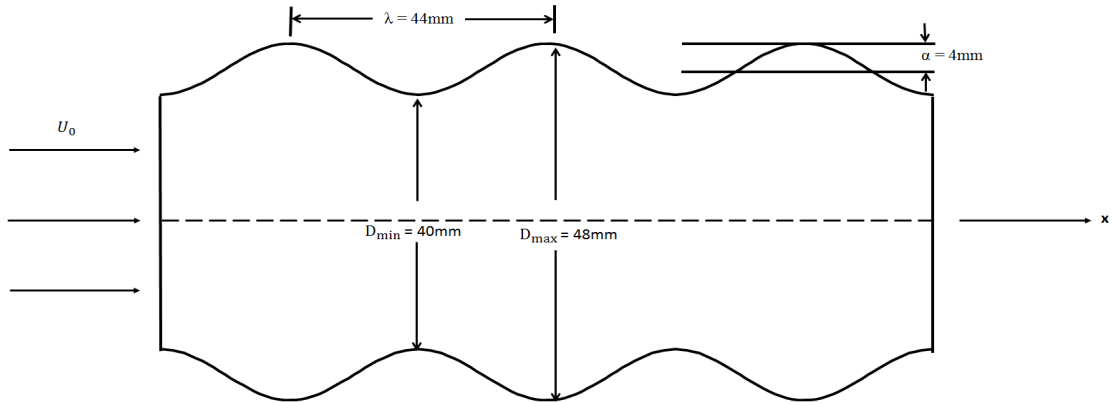
$$r = a \left[1 - \sin \frac{\pi}{2} \left(1 + \frac{4x}{\lambda} \right) \right] + \frac{D_{min}}{2}. \quad (2.1)$$



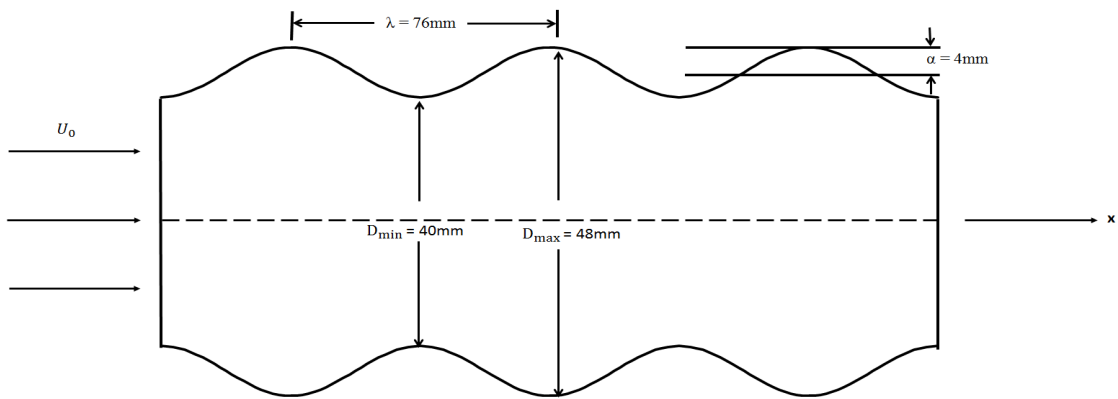
(a) A typical smooth pipe



(b) $\lambda/a = 1$



(c) $\lambda/a = 11$



(d) $\lambda/a = 19$

Figure 2.1: A simplified diagram of a part of geometry under study (Total length is 7λ)

U_0 is the constant velocity at the inlet, x is the axial distance, D_{min} is the minimum diameter of the wavy wall, D_{max} is the maximum diameter of the wavy wall and r is the radial distance of the pipe. Water is the fluid medium considered with fluid properties assumed to be density $\rho = 1000 \text{ kg/m}^3$, kinematic viscosity $\nu = 1 \times 10^{-6} \text{ m}^2/\text{s}$ and dynamic viscosity, $\mu = 1 \times 10^{-3} (\text{N.s})/\text{m}^2$, and the Reynolds numbers 10 000, 12 500, 15 000, 17 500 and 20 000 are simulated using the two turbulence models ($k - \epsilon$ and $k - \omega$). Computations were done utilizing the commercially accessible CFD software ANSYS Fluent.

Chapter 3

Governing Equations

This section presents the governing equations utilized in the current investigation. Presented in this section is the governing equations and boundary conditions for our investigation. The domain is discretized into tiny size cells to form a volume mesh or grid and algorithms in ANSYS FLUENT are utilized to solve. The governing equations for viscous flow are solved numerically.

Turbulent flows are hard to simulate because of their randomness. There are fundamentally three approaches to simulate turbulent flow: Direct Numerical Simulation (DNS), Reynolds Averaged Navier-Stokes (RANS) and Large-Eddy Simulation (LES) models. The additional details we would like to attain from a simulation, the more expensive is the computational value. DNS and LES provide important details, however need extremely massive times for the flow simulations, that are of importance to us. Given our intentions and the accessible computational power, we are going to use Reynolds Averaged Navier-Stokes models (particularly the $k - \epsilon$ and $k - \omega$) for our simulations. The Reynolds Averaged Navier-Stokes equations are time averaged equations of motion for fluid flow. The averaging method carries new unknown conditions into the Navier-Stokes equations. Accordingly, extra (closure) equations are required in order to solve the system of equations. These equations are determined by means of taking higher-order moments of the averaged Navier-Stokes equations and creating further assumptions supported by the understanding of the characteristics of the turbulent flow. The procedure brings about a modified

set of equations that is computationally more affordable to solve. Below, the equations, which outline the $k - \epsilon$ model and also the $k - \omega$ models are provided.

The numerical simulation of the incompressible flow, single phase, and isothermal is achieved using CFD, via the commercial software program ANSYS FLUENT.

3.1 Basic Model and Assumptions

3.1.1 Assumptions

The numerical model depends on the following assumptions:

- the flow is three- dimensional, steady and turbulent;
- the fluid is incompressible fluid;
- the flow is single phase, e.g. the impacts of dirt particles and additionally water vapor were ignored;
- the air properties are constant,
- the velocity is constant over the pipe inlet;
- flow is symmetric about the axis of the pipe.

3.1.2 Governing Equations

In the current problem, the fluid flow is three dimensional; that is, each of the three feasible velocity components (x , y and z) remain and all three possible constituents rely on the three coordinates of Cartesian geometry. The governing equations describing the development of flow inside the pipe are given below:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0. \quad (3.1)$$

x -momentum equation is

$$\begin{aligned} \rho \left[\frac{\partial}{\partial x}(u^2) + \frac{\partial}{\partial y}(uv) + \frac{\partial}{\partial z}(uw) \right] &= -\frac{\partial \rho}{\partial x} + \frac{\partial}{\partial x} \left(\mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu \frac{\partial u}{\partial y} \right) \\ &+ \frac{\partial}{\partial z} \left(\mu \frac{\partial u}{\partial z} \right), \end{aligned} \quad (3.2)$$

y -momentum equation is

$$\begin{aligned} \rho \left[\frac{\partial}{\partial x}(vu) + \frac{\partial}{\partial y}(v^2) + \frac{\partial}{\partial z}(vw) \right] &= -\frac{\partial \rho}{\partial x} + \frac{\partial}{\partial x} \left(\mu \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu \frac{\partial v}{\partial y} \right) \\ &+ \frac{\partial}{\partial z} \left(\mu \frac{\partial v}{\partial z} \right), \end{aligned} \quad (3.3)$$

z -momentum equation is

$$\begin{aligned} \rho \left[\frac{\partial}{\partial x}(wu) + \frac{\partial}{\partial y}(wv) + \frac{\partial}{\partial z}(w^2) \right] &= -\frac{\partial \rho}{\partial x} + \frac{\partial}{\partial x} \left(\mu \frac{\partial w}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu \frac{\partial w}{\partial y} \right) \\ &+ \frac{\partial}{\partial z} \left(\mu \frac{\partial w}{\partial z} \right). \end{aligned} \quad (3.4)$$

where u , v , and w are the velocity components in x , y and z directions respectively. ρ is the fluid density, and μ the molecular viscosity.

3.2 Turbulence Models

Variations in the velocity field join transported quantities like energy and momentum and induce the transported quantities to fluctuate likewise. These variations can be of an extremely small scale and consequently can make an amazingly substantial computational costs for practical engineering calculations. A changed arrangement of equations that need considerably less computational cost are used. This is accomplished through time-averaging the instantaneous governing equations which at that point include extra unknown variables. These unknown variables are solved through turbulence models.

Not a single turbulence model can be generally implemented to all circum-

stances. When selecting a turbulence model the following considerations must be taken; physics incorporated in the flow; precision level; and computation assets accessible. In this chapter, two turbulence transport models will be investigated and discussed. Both the standard $k - \epsilon$ turbulence model along with the standard $k - \omega$ turbulence model were discussed in this study.

The equations of motion are averaged with regard to time and these turbulent transport models predict the impact of turbulence on the time averaged mean motion. Considering all of the terms presently in the equations of motion are immediate values, they are substituted with the total time-mean quantity and a fluctuating quantity. A new arrangement of equations of motion with a similar arrangement are acquired with the inclusion of these quantities.

Momentum in x -direction

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \bar{u}) + \frac{\partial}{\partial x}(\rho \bar{u}u) + \frac{\partial}{\partial y}(\rho \bar{v}u) + \frac{\partial}{\partial z}(\rho \bar{w}u) &= -\frac{\partial \bar{p}}{\partial x} + \mu \left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} + \frac{\partial^2 \bar{u}}{\partial z^2} \right) \\ &- \frac{\partial}{\partial x_i}(\rho \overline{u' u'_i}), \end{aligned} \quad (3.5)$$

Momentum in y -direction

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \bar{v}) + \frac{\partial}{\partial x}(\rho \bar{u}v) + \frac{\partial}{\partial y}(\rho \bar{v}v) + \frac{\partial}{\partial z}(\rho \bar{w}v) &= -\frac{\partial \bar{p}}{\partial y} + \mu \left(\frac{\partial^2 \bar{v}}{\partial x^2} + \frac{\partial^2 \bar{v}}{\partial y^2} + \frac{\partial^2 \bar{v}}{\partial z^2} \right) \\ &- \frac{\partial}{\partial x_i}(\rho \overline{v' u'_i}), \end{aligned} \quad (3.6)$$

Momentum in z -direction

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \bar{w}) + \frac{\partial}{\partial x}(\rho \bar{u}w) + \frac{\partial}{\partial y}(\rho \bar{v}w) + \frac{\partial}{\partial z}(\rho \bar{w}w) &= -\frac{\partial \bar{p}}{\partial z} + \mu \left(\frac{\partial^2 \bar{w}}{\partial x^2} + \frac{\partial^2 \bar{w}}{\partial y^2} + \frac{\partial^2 \bar{w}}{\partial z^2} \right) \\ &- \frac{\partial}{\partial x_i}(\rho \overline{w' u'_i}). \end{aligned} \quad (3.7)$$

Continuity

$$\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0. \quad (3.8)$$

The momentum equations have the term $-\frac{\partial}{\partial x_i}(\rho \overline{u'_j u'_i})$ presented in compact tensor notation. These terms symbolize the impact that turbulent motion has on the time-mean quantities recognized as Reynolds stresses which are very often varying velocity elements.

The concept of eddy-viscosity projected by Boussinesq would then be able to be used to the motion equations. The extra turbulent stresses are presumed to be in proportion to the mean-velocity gradients. The stresses in tensor notation can be connected thus.

$$\overline{u'_j u'_i} = \frac{\mu_i}{\rho} \left(\frac{\partial \overline{u_j}}{\partial x_i} + \frac{\partial \overline{u_i}}{\partial x_j} \right) - \frac{2}{3} k \delta_{ji}. \quad (3.9)$$

Where μ_i , is the eddy viscosity, k is the kinetic energy, and δ_{ji} is the Kronecker delta. This method is utilized in the $k - \epsilon$ as well as $k - \omega$ turbulence models. The benefits of this method is that computational expenses are moderately low due to the computational ability required to compute the turbulent viscosity.

In x -direction, $\frac{\partial}{\partial x_i}(\rho \overline{u' u'_i})$ extends to $\frac{\partial}{\partial x}(\rho \overline{u' u'}) + \frac{\partial}{\partial y}(\rho \overline{u' v'}) + \frac{\partial}{\partial z}(\rho \overline{u' w'})$.

The eddy viscosity is a property of the flow, not the fluid. By substituting, the varying quantities are eliminated as mean quantity gradients and eddy viscosity define the turbulent dispersion and the equations of motion are depicted thus.

Momentum in x -direction

$$\frac{\partial}{\partial t}(\rho \overline{u}) + \frac{\partial}{\partial x}(\rho \overline{u u}) + \frac{\partial}{\partial y}(\rho \overline{v u}) + \frac{\partial}{\partial z}(\rho \overline{w u}) = -\frac{\partial \overline{p}}{\partial x} + (\mu + \mu_i) \left(\frac{\partial^2 \overline{u}}{\partial x^2} + \frac{\partial^2 \overline{u}}{\partial y^2} + \frac{\partial^2 \overline{u}}{\partial z^2} \right), \quad (3.10)$$

Momentum in y -direction

$$\frac{\partial}{\partial t}(\rho \bar{v}) + \frac{\partial}{\partial x}(\rho \bar{u} \bar{v}) + \frac{\partial}{\partial y}(\rho \bar{v} \bar{v}) + \frac{\partial}{\partial z}(\rho \bar{w} \bar{v}) = -\frac{\partial \bar{p}}{\partial y} + (\mu + \mu_i) \left(\frac{\partial^2 \bar{v}}{\partial x^2} + \frac{\partial^2 \bar{v}}{\partial y^2} + \frac{\partial^2 \bar{v}}{\partial z^2} \right), \quad (3.11)$$

Momentum in z -direction

$$\frac{\partial}{\partial t}(\rho \bar{w}) + \frac{\partial}{\partial x}(\rho \bar{u} \bar{w}) + \frac{\partial}{\partial y}(\rho \bar{v} \bar{w}) + \frac{\partial}{\partial z}(\rho \bar{w} \bar{w}) = -\frac{\partial \bar{p}}{\partial z} + (\mu + \mu_i) \left(\frac{\partial^2 \bar{w}}{\partial x^2} + \frac{\partial^2 \bar{w}}{\partial y^2} + \frac{\partial^2 \bar{w}}{\partial z^2} \right). \quad (3.12)$$

Continuity

$$\frac{\partial}{\partial x}(\rho \bar{u}) + \frac{\partial}{\partial y}(\rho \bar{v}) + \frac{\partial}{\partial z}(\rho \bar{w}) = 0. \quad (3.13)$$

This arrangement of equations varies from the first instantaneous equations only because of the momentum coefficients. The influence of turbulence on the time-mean quantities is represented by the new terms in the momentum equations. The following sections outline the turbulence models that will be utilized in the current study.

3.2.1 $k - \epsilon$ Turbulence Model

The $k - \epsilon$ turbulence model has evolved into a standout amongst the extensively utilized turbulence models as it gives vigor, prudence and reasonable precision for an extensive variety of turbulent flows. Upgrades have been made to the standard model which enhances its execution and two derived forms are accessible in Fluent; the renormalization group (RNG) model. The standard, RNG and realizable models have comparable order with transport equations for k and ϵ . These two transport equations freely determine the turbulent velocity and length scales. The primary contrasts among the three models are as per the following;

- The turbulent Prandtl Numbers guiding the turbulent diffusion of k and ϵ
- The generation and destruction terms in the equation for ϵ
- The means of computing turbulent viscosity

3.2.1.1 Standard $k - \epsilon$ Turbulence Model

This unique model was originally projected by Launder and Spalding [31]. For Standard $k - \epsilon$ turbulence model the transport equation for k is obtained from the same equation, but the transport equation for ϵ was derived by applying natural logic and as a result very much alike to the mathematically acquired transport equation of k , although it is not accurate. The turbulent kinetic energy k , and its rate of dissipation ϵ , for this model are acquired as per the following equations.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial y}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_i}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + \rho \epsilon + S_k. \quad (3.14)$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial y}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_i}{\sigma_k} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} G_k - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon. \quad (3.15)$$

In which G_k denotes the generation of turbulent kinetic energy that emerges because of mean velocity gradients. S_ϵ and S_k are source terms outlined by the user.

Constants $C_{1\epsilon}$ and $C_{2\epsilon}$ have been resolved experimentally and are considered to posses the following values;

$$C_{1\epsilon} = 1.44, \quad C_{2\epsilon} = 1.92,$$

σ_k are σ_ϵ turbulent Prandtl numbers for the turbulent kinetic energy and its dissipation rate. They likewise have been obtained experimentally and are characterized as per the following:

$$\sigma_k = 1.0, \quad \sigma_\epsilon = 1.3.$$

At every point the eddy (or turbulent) viscosity is identified with the local values of turbulent kinetic energy and its dissipation rate by:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}, \quad (3.16)$$

where C_μ is constant and $C_\mu = 0.09$.

The term for the generation of turbulent kinetic energy G_k is familiar in most of the turbulent models studies and is characterised as

$$G_k = -\overline{\rho u'_i u'_j} \frac{\partial u_j}{\partial x_i}. \quad (3.17)$$

The modulus of mean rate-of-strain tensor, S is defined as

$$S = \sqrt{2S_{ij}S_{ij}}. \quad (3.18)$$

3.2.2 $k - \omega$ Turbulence Model

The next turbulence model to be examined is the $k - \omega$ turbulence model. This model has two variations that will both be characterised in this section; the Standard $k - \omega$ turbulence model, and the Shear Stress Transport (SST) model. These two turbulence models utilize comparable transport equations for k and ω but the Shear Stress Transport model varies from the standard model as follows:

- There is a continuous alteration from the standard $k - \omega$ model in the inward region of the boundary layer to the $k - \epsilon$ model in the external portion of boundary layer.
- As a means to account for the transport effects of the main turbulent shear stresses the SST model includes an altered turbulent viscosity equation

The sections following clarify these distinctions, introduce the transport equations and demonstrate the techniques of solving for turbulent viscosity and all

constants.

3.2.2.1 Standard $k - \omega$ Turbulence Model

FLUENT utilizes a standard $k - \omega$ model created by Wilcox [32] that was developed to improve calculate low-Reynolds number effects, compressibility, and shear flow spreading. This standard model is an experimentally founded model with transport equations for turbulent kinetic energy k and its particular dissipation rate ω . This model has been changed several times trying to enhance its precision and therefore the transport equations utilized in FLUENT for Wilcox's model are as per the following.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial y}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k. \quad (3.19)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial y}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + S_\omega. \quad (3.20)$$

Where G_k depicts the production of turbulent kinetic energy that emerges because of mean velocity gradients and G_ω is production of ω which are described in the same way as the $k - \epsilon$ model. Y_k and Y_ω describe the dissipation of k and ω because of turbulence. Y_k and Y_ω are the turbulent Prandtl number for k and ω respectively and S_ω and S_k are source terms outlined by the user. A damping coefficient (α^*) is used to define the turbulent viscosity;

$$\mu_t = \alpha^* \frac{\rho k}{\omega}. \quad (3.21)$$

The correlation of low Reynolds number is acquired from this constant α^* which damps the turbulent viscosity. This constant is defined by,

$$\alpha^* = \alpha_\infty^* \left(\frac{\alpha_0^* + Re_t/R_k}{1 + Re_t/R_k} \right), \quad (3.22)$$

Where $e_t = \frac{\rho k}{\mu_\omega}$, $R_k = 6$, $\alpha_0^* = \frac{\beta_i}{3}$ and $\beta_i = 0.072$. For the high Reynolds number order of the Standard $k - \omega$ model, $\alpha^* = \alpha_\infty^* = 1$.

The expression for the dissipation of k because of turbulence, Y_k is characterized

as

$$Y_k = \rho\beta^* f_{\beta^*} k\omega. \quad (3.23)$$

And the expression for the dissipation of ω because of turbulence, Y_ω is characterized as

$$Y_\omega = \rho\beta f_\beta \omega^2. \quad (3.24)$$

The $k - \omega$ turbulence model created by Wilcox [32] has revealed similar conclusions for mixing layers, wakes and different jet models which make it appropriate for free shear flows and wall bounded flows.

Chapter 4

Numerical Simulation

4.1 Computational Fluid Dynamics

This section details the basics of Computational Fluid Dynamics (CFD), history on CFD and the finite volume method (FVM), the solver parameters for the commercial CFD program and the post-processing technique in the following areas:

4.1.1 CFD Background

Computational Fluid Dynamics (CFD) utilizes numerical methods to analyze and solve problems that include fluid flows. The governing equations considered before have been solved with regard to the given boundary conditions employing the FVM as applied in the commercial CFD software FLUENT.

CFD analysis has three parts, which are; the pre processor, the solver, and the post processor which is characterized as a application that convert input data to generate output that is utilized as an input to the processor. In the current investigation, ANSYS fluent was utilized to make the geometry of the structure being studied and partition the domain into components which entirely form the mesh or grid. ANASYS FLUENT is an efficient Computational Fluid Dynamic program for laminar and turbulent flows in complex two and three dimensional configurations. This program is utilized in research activities and

industrial operations in various areas associated to energy production (gas and coal combustion, nuclear thermal-hydraulics, turbo machinery, heating, ventilation and air conditioning). The arrangement of the flow is afterward explained at the center of each of these volumes by means of the solver. The pre processor likewise characterizes common surface types (for instance-inlet and outlet regions) to areas of the domain.

The pre-processor discretizes the differential equations changing them into algebraic equations that can be solved numerically utilizing the finite volume method and this method is explained later in this section in more details. Lastly the post processor is a tool that gives the explanation and understanding of the solution in the type of plots, charts and graphs. FLUENT is employed for both the solver and the current study. Microsoft Excel is likewise utilized as a post processor for preparing charts and graphs to be displayed in this report.

Jones and Whittle [33] explain in details the use of CFD to building geometry construction, including techniques used.

4.1.2 Pre-Processor

The pre-processor comprises of the inputs of a flow problem to a CFD code by ways of a user-friendly interface and the following change of this input into a form appropriate to be used by the solver. In the current study the pre-processor is incorporated in ANSYS fluent and the operations at the pre-processing stage include:

- Description of the geometry of the area of interest: the computational domain
- Mesh formation on the sub division of the domain into a various smaller, not overlapping sub-domain: a grid of cells for control volumes ,
- Choice of the physical and chemical phenomena that should be modelled
- Definition of fluid properties

- Stipulation of suitable boundary conditions at cells which concur with or make contact with the domain boundary

4.1.3 Solver (FLUENT Solver)

The numerical methods that form the basics of the fluent-solver carry out the following steps:

- Estimation of the unknown flow variables by way of simple function
- Discretization through substitution of the estimations into the governing flow equations and subsequent mathematical operations
- Solution of the algebraic equations.

ANSYS FLUENT is well suited for two and three-dimensional calculations of steady or transient single-phase, incompressible, laminar or turbulent flows. It supports two Reynolds Averaged Stoke (RANS) models: The standard $k - \epsilon$ model, the lauder Sharman model and a second Moment closure (LRR) model. It also contains the LES Smagorinsky and dynamic models. Some of the turbulence models which are also implemented in fluent are The SST model, elliptic relaxation models $v^2 - f$, the SSG model and the scalable wall function.

The flow solver is based on a finite volume method, with a completely collected arrangement for all variables. The time discretization is similar to the method used in other commercial codes. It is based on a predictor-corrector scheme for the Navier- Stokes equations.

An important asset of ANSYS FLUENT is it relies on its ability to deal with any kind of mesh (hybrid, containing arbitrary interfaces and any type of cell).

4.1.4 Post-Processor

Like in pre-processing a vast measure of progress has as of late occurred in the post-processor field. Because of the increased recognition of engineering

workstations, several of which have remarkable graphics capacity, the main CFD packages are presently furnished with flexible data visualization devices. This involve:

- Domain geometry and mesh presentation
- Vector plots
- Shaded and line contour plots
- Particle tracing
- View control (translation, revolution, scaling, and so on.)
- Color postscript output

4.2 The Finite Volume Method

The method relies on:

- Dividing the domain into control volumes.
- Formal integrations of the governing equations of fluid flow past all the control volumes of the solution domain.
- Discretization includes the replacement of a diversity of finite-difference kind of approximations for the terms in the integrated equation portraying flow processes, for example, diffusion, sources and convection. This changes the integral equations into an arrangement of algebraic equations.
- Solution of the algebraic equations by an iterative method.
- The preservation of a common flow variable ϕ . For instance a velocity element inside a finite control volume can be signified as a harmony between the different procedures inclined to increase or decrease it.

In this chapter, (i, j) refer to the cell on either side of a face while the lower case letters reference the usual tensor quantities

$$\int \frac{\partial \rho \phi}{\partial t} dV + \int \frac{\partial}{\partial x_j} (\rho u_j \phi) dV = \int \frac{\partial}{\partial x_j} \left[\Gamma \frac{\partial \phi}{\partial x_j} \right] dV + \int S dV. \quad (4.1)$$

Where “ S ” represents the source term, “ V ” the volume of the cell and “ Γ ” the diffusion coefficient. Using Gauss theorem the volume integrals of the divergence can be transformed into surface integrals, which can be written as:

$$\int \frac{\partial \rho \phi}{\partial t} dV + \int \rho \phi u_j n_j dA = \int \Gamma \frac{\partial \phi}{\partial x_j} n_j dA + \int S dV. \quad (4.2)$$

Where “ A ” is the area of the face and “ n_j ” is the face normal vector. The volume integrals are approximated by the product of the cell center S_I and the cell volume V

$$\int S dV = S_I V. \quad (4.3)$$

4.2.1 Face integrals

The surface integrals can be approximated by the mid-point rule, the product of the face center value and the face. In the collocated arrangement, an interpolation is required in order to obtain the centre, since all the variables are stored at the cell centre. In ANSYS FLUENT the convection can be calculated either by using an upwind differencing scheme, the value at the cell Centre to dynamically switch from (CDS) or a central differencing scheme (CDS). The code has also a slope test based on the product of the gradients at the cell center to dynamically switch from CDS to USD. Using the upwind scheme, the value at the face can be obtained as:

$$\begin{aligned} \phi_F &= \phi_I & \text{if } u_i n_{iF}, \\ \phi_F &= \phi_j & \text{if } u_i n_{iF}. \end{aligned} \quad (4.4)$$

This scheme is robust and stable but introduces additional numerical diffusion

which can become large if the grid is coarse. For a centered scheme, the value at the face can be computed as:

$$\phi_F = \alpha\phi_I + (1 - \alpha)\phi_j + \left[\frac{1}{2} \left(\frac{\partial\phi}{\partial x_j} I_I + \frac{\partial\phi}{\partial x_j} I_j \right) O_{F_j} \right], \quad (4.5)$$

With $\alpha = \frac{F_{J'}}{I'J'}$ The last term is added for non-orthogonal grids, where the center of the face does not lie in the mid-point between the cell centers. The diffusion integral can be computed as:

$$\int \frac{\partial\phi}{\partial x_j} n_j dA = \sum_{Neigh} \Gamma \frac{\partial\phi}{\partial x_j} n_j A, \quad (4.6)$$

with a linear approximation for the gradient the face center, the diffusion is computed as:

$$\sum_{Neigh} \Gamma \frac{\partial\phi}{\partial x_j} n_j A = \sum_{Neigh} \Gamma \frac{\phi_{j'} - \phi_{I'}}{IJ_j} n_j A. \quad (4.7)$$

The values of $\phi_{j'}$ and $\phi_{I'}$ can be calculated by using the gradient at the cell center:

$$\phi_{I'} = \phi_I + \frac{\partial\phi}{\partial x_j} I_I I' I_j. \quad (4.8)$$

4.2.2 Gradient Reconstruction

The calculation of the gradients is achieved by an iterative solver in which the gradient is expressed as:

$$\frac{\partial\phi}{\partial x_j} I_I = \frac{1}{V} \sum_{Neigh} \int \phi n_j dA. \quad (4.9)$$

The surface integral can be approximated using the midpoint rule so that it becomes:

$$\frac{\partial\phi}{\partial x_j} I_I = \frac{1}{V} \sum_{Neigh} \phi_F A_F n_j. \quad (4.10)$$

To obtain the values of ϕ_F Taylor series expansion can be applied to obtain:

$$\phi_F = \phi_0 + OF_j \frac{\partial \phi}{\partial x_j} I_0 + O. \quad (4.11)$$

The value of ϕ_0 can be obtained by a linear interpolation and the gradient at the same point, from an averaged between the values of $\frac{\partial \phi}{\partial x_j} I_I$ and $\frac{\partial \phi}{\partial x_j} I_j$. Finally the system to solve can be written as:

$$\frac{\partial \phi}{\partial x_j} I_I = \frac{1}{V} \sum_{Neigh} \left\{ \alpha \phi_I + (1 - \alpha) \phi_j + \frac{1}{2} OF_j \left[\frac{\partial \phi}{\partial x_j} I_I + \frac{\partial \phi}{\partial x_j} I_j \right] \right\} A_F n_j. \quad (4.12)$$

4.3 Time Discretization

The time discretization in FLUENT is achieved through a fractional step scheme that can be associated the SIMPLEC method.

The solution algorithm consist a prediction-correction method. At first the momentum equation is solved using an explicit pressure gradient from the previous time step. With $u_i^n = pu_i^n$ being the momentum at time step n the system to solve at the first step of the method is:

$$\frac{Q_i^* - Q_i^n}{\Delta t} + \frac{\partial}{\partial x_j} \left(u_i^* Q_j^n - \mu \frac{\partial u_i^*}{\partial x_j} \right) = -\frac{\partial P^n}{\partial x_i} + S^*. \quad (4.13)$$

Where S includes all the source terms that can be made implicit or explicit, $S = A_i^n + B_i^n u_i$ and Δt is the time step. After this prediction step a new velocity field is obtained (denoted by $(*)$) which is usually not divergence free. The second step consists of calculating the pressure gradient in order to satisfy the continuity equation. By taking the divergence of the momentum equation, the Poisson equation for the pressure can be written as:

$$\frac{\partial}{\partial x_j} \left(\Delta t \frac{\partial (P^{**} - P^*)}{\partial x_j} \right) = \frac{\partial \phi_i^*}{\partial x_j}. \quad (4.14)$$

Finally, once the updated pressure (P^{**}) has been obtained, the velocity field is corrected. This is done by neglecting convection and diffusion variations:

$$Q_i^{**} - Q_i^* = -\Delta t \frac{\partial}{\partial x_i} (P^{**} - P^*). \quad (4.15)$$

The velocities and the pressure are updated, that is $Q^{n+1} = Q^{**}$ and $P^{n+1} = P^{**}$ when a turbulence model is used, the resolution of the turbulence variables takes place after the velocities are computed. The dependence of other variables is fully explicit so that each equation is solved separately.

4.4 Boundary Conditions

For the resolution of discretized equations described previously, the boundary conditions have to be prescribed for any variable ϕ .

4.4.1 Inlet

At the inlet a Dirichlet condition is prescribed for all transport variables (velocity, scalars, turbulent variables) so the values for ϕ_{inlet}^{n+1} and Q_{inlet}^{n+1} are prescribed by the user. A homogeneous Neumann condition (zero flux) is imposed on the pressure. The convection term $Q_j^n n_j \phi$ is calculated directly from the prescribed values. For the diffusion terms, the boundary value is calculated as:

$$\left(\Gamma \frac{\partial \phi}{\partial x_j} n_j \right)_{inlet} = \Gamma \frac{\phi_{inlet}^{n+1} - \phi_{I'}^*}{I' F_j n_j}. \quad (4.16)$$

For the source terms, the recommended value ϕ_{inlet}^{n+1} is used as a boundary value. The pressure gradient normal to the face is prescribed as zero although as extrapolation from the previously obtained results is possible. The term $(\Delta t \frac{\partial \delta P}{\partial x_j} n_j)$ is set to zero in the Poisson equation and the term on the right hand side is calculated as:

$$Q_j^* n_j = \rho_{inlet}^{n+1} u_{j(inlet)}^{n+1} n_j. \quad (4.17)$$

4.4.2 Outlet

For the outlet, a homogeneous Neumann condition is imposed on the velocity, scalars and turbulent variables. For the pressure, a Dirichlet condition is used on P_{outlet}^{n+1} . The boundary values for the diffusion terms to zero, and for the source terms a first order approximation is used to set $\phi = \phi_{I'}$. The Dirichlet condition is also used in the Poisson equation. In the momentum correction equation, the boundary value for δP is set to zero.

4.4.3 Walls and Symmetries

For the walls and symmetry faces, a zero mass flux is imposed and both Dirichlet and Neumann conditions can be applied to scalars. For the tangential velocity, a homogeneous Dirichlet boundary condition is used at the wall whereas at the symmetry faces is set to zero although it can also be computed via an explicit extrapolation of the value at the boundary cell.

For convection terms, the boundary value of the flux is set to zero. For diffusion terms when a Neumann condition is applied to the variable ϕ ; the value prescribed is used directly. If a Dirichlet condition is applied, the boundary value is calculated in the same way as inlet.

If the variable has a Dirichlet boundary condition and a source term requiring the gradient of the variable, the prescribed value is used as boundary value $\phi_b = \phi$. If a Neumann condition is used, an extrapolated value from the boundary cell is used with a first order approximation. For the velocity component normal to the wall, the boundary value is set to ensure zero mass flow.

For the pressure gradient calculation, if the flux of the variable at the face is prescribed, the boundary value is:

$$\phi_b = \phi_{I'} + \Gamma F \frac{\partial \phi}{\partial x_j} n_j I_b. \quad (4.18)$$

To solve the Poisson equation, the values of $\Delta t \frac{\partial \delta P}{\partial x_j} n_j$ and $Q_j^* n_j$ for the momen-

tum correction equation, the boundary value for δP is obtained from the cell value from a first order approximation using the fact that $\Delta t \frac{\partial \delta P}{\partial x_j} n_j = 0$.

The value for the boundary conditions at the wall is prescribed for all velocities and turbulent variables. The way this boundary condition is treated depends on the turbulence model used. Many turbulence models have been designed under a local equilibrium assumption and use the universal logarithmic law to bridge the viscous sub layer, hence solving for the flow outside the buffer layer. To prescribe the velocity at the wall, the total shear stress is required:

$$\tau_\omega = \rho u^* u_k, \quad (4.19)$$

With u^* is the friction velocity and $u_k = \sqrt{k/C_\mu^{1/2}}$. In the code, the shear is calculated by using the wall function approach. Defining the tangential velocity at the wall as $u_j^{tg} = u_j - u_i n_i n_j$. The shear stress can be computed from:

$$u_k = C_\mu^{-1/4} k^{1/2}, \quad (4.20)$$

$$u^* = \frac{\sqrt{u_j^{tg} u_j^{tg}}}{\frac{1}{k} \ln(y^+) + C}. \quad (4.21)$$

With $k = 0.42$ and $C = 5.2$.

For the turbulent variables, the boundary conditions are obtained from:

$$\frac{\partial k}{\partial y} = 0, \quad (4.22)$$

$$\frac{\partial \epsilon}{\partial y} = \frac{u_k^\epsilon}{k y^2}. \quad (4.23)$$

In the case of $k - \epsilon$ model and for the second moment closure the conditions are:

$$\frac{\partial u'_j u'_j}{\partial x_j} n_j = 0, \quad (4.24)$$

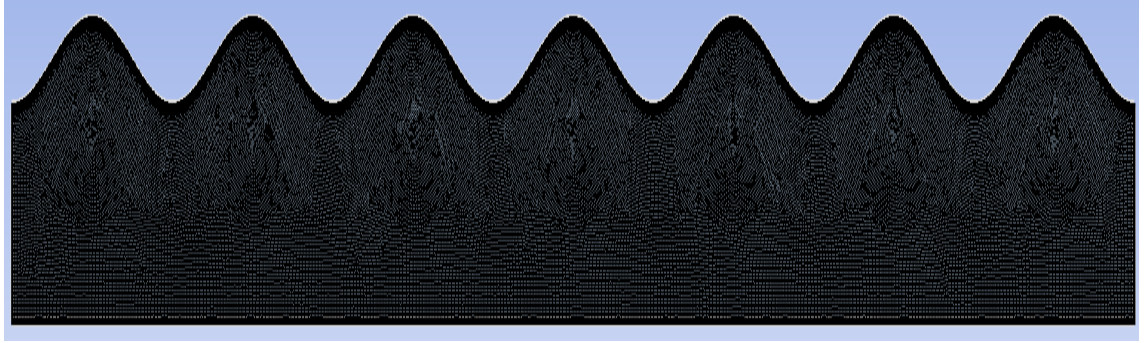
$$u_1' u_2' = u^* u_k, \quad (4.25)$$

$$u_1' u_3' = u_2 u_3 = 0, \quad (4.26)$$

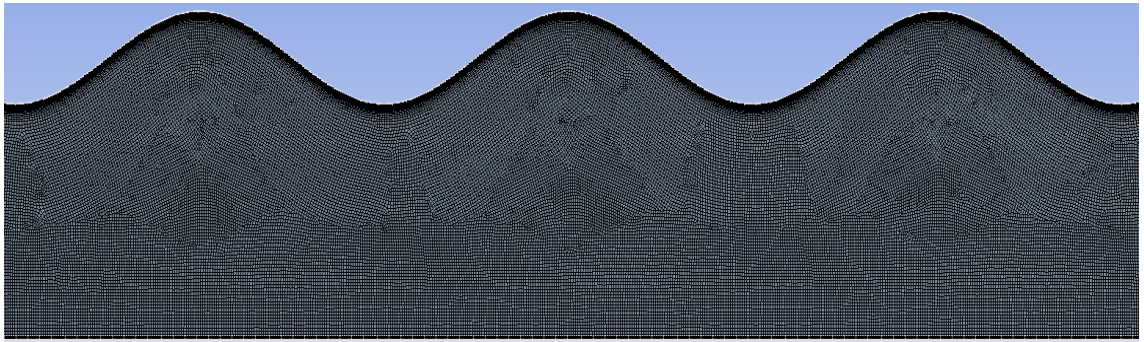
$$\frac{\partial \epsilon}{\partial y} = \frac{u_k^\epsilon}{ky^2}. \quad (4.27)$$

This is in a local frame where x_2 is normal to the wall.

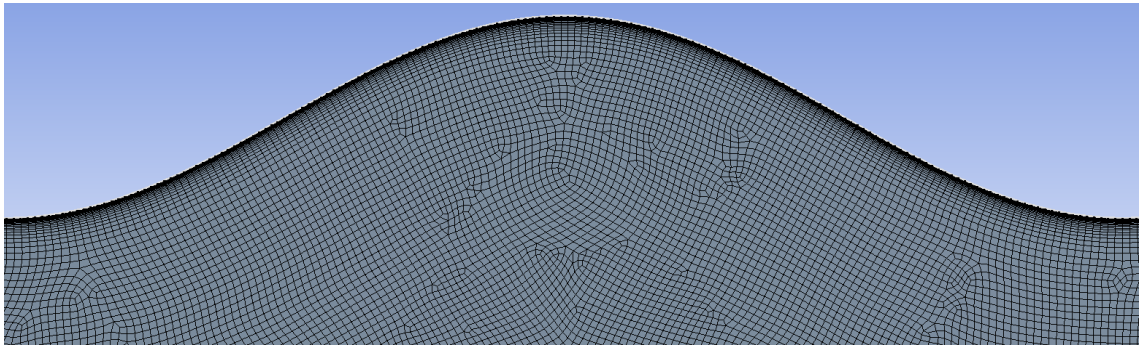
Figure 4.1 exhibits a simplified description of the mesh distribution inside the corrugated pipe. We observe that the areas close to the corrugation and near the wall need finer meshes than the region close to the center of the pipe.



(a) full pipe



(b) zoomed to 3λ



(c) inside the corrugation cavity

Figure 4.1: Mesh distribution (a) in the full pipe (b) zoomed to 3λ and (c) inside the corrugation cavity

Chapter 5

Results and Discussions

In this chapter, the results of the current study are presented and discussed. Before the results are presented, the methodology, modelling and boundary conditions are validated. Validation is carried out by using the following methods.

1. Checking the mass flow rate at the inlet and outlet,
2. Carrying out a grid independence study,
3. Comparing our results for an already existing geometry and available results.

5.1 Conservation of Mass

At the end of each computation, mass flow at the inlet and outlet planes are computed and checked. Since for an incompressible flow mass is conserved, the amount of mass of fluid going into the computational domain must be the same as the mass of the fluid going out of the domain. A tolerance limit of 0.01% was fixed for determining the convergence.

5.2 Grid Independence

Grid independence study is carried out to remove or decrease the impact of the mesh size on the computational outcomes. Grid convergence is the term used to describe the improvement of results by using successively smaller cell sizes for

the calculations. A calculation should approach the correct answer as the mesh becomes finer, hence the term grid independence. It is forever an acceptable practice to adhere to this for all geometries, which is monotonous. The grid independence for a particular geometry will be relevant just for that geometry.

In our case repeated computations were done for both coarse and fine grid sizes and the grid size was found to have no effect or small effect on the results varying less than 1%. Therefore, the fine grid size was chosen. In the grid size a mesh size of 90 000 to 120 000 elements were computed but this did not have any effect in the accuracy of the solutions. Table 5.1 shows grid sizes for each geometry investigated in this paper.

Geometry	Grid Elements	Grid Nodes
$\lambda/\alpha = 1$	110 730	110 798
$\lambda/\alpha = 11$	97 618	101 181
$\lambda/\alpha = 19$	91 265	95 627

Table 5.1: Grid sizes for each geometry

However Table 5.2, shows the results of computations carried out using various grid sizes for the geometry with $\lambda/\alpha = 1$ at Re 20 000 using $k - \epsilon$ model. As it can be seen from Table 5.2, the skin friction factor for the geometry $\lambda/\alpha = 1$ did not vary significantly (less than 0.01%) for the grid sizes of 110 730 and 116 260 grid elements in the computational domain. Hence 110 730 grid elements were chosen for further analysis for this geometry.

Grid Elements	Grid Nodes	Friction Factor
90 190	90 360	0.0725822
96 390	96 512	0.0712428
101 640	101 752	0.0701899
110 730	110 798	0.0700358
116 260	116 435	0.0700298

Table 5.2: Grid independence study for the geometry $\lambda/\alpha = 1$ at Re 20 000 using $k - \epsilon$ model

5.3 Smooth Wall Validation

To authenticate turbulent flow simulation in pipes with smooth walls, a post process is carried out at the end of every simulation. The fluid properties assumed are density $\rho = 1000 \text{ kg/m}^3$, dynamic viscosity, $\mu = 1 \times 10^{-3}$ and kinematic viscosity $\nu = 1 \times 10^{-6} \text{ m}^2/\text{s}$, and the Reynolds numbers 10 000, 12 500, 15 000, 17 500 and 20 000 are simulated using the two turbulence models ($k - \epsilon$ and $k - \omega$). Computations were carried out using ANSYS Fluent. The average velocity V_{avg} , Reynolds number Re , and Darcy-Weisbach friction factor f are calculated as follows:

$$V_{avg} = \int_0^R \frac{U_1 2\pi r}{\pi R^2} dr, \quad \text{Re} = \frac{2R\rho V_{avg}}{\mu}, \quad f = \frac{2D\Delta P}{\rho V_{avg}^2 L}.$$

Where ΔP is the recommended pressure change between inlet and outlet, D is the diameter of the pipe, and L is the length of the pipe segment incorporated in the computational domain.

For each Reynolds number, we calculated friction factors for every geometry plus the friction factor acquired from Blasius Correlation and the friction factor acquired from the Moody diagram [4, 34].

$$\text{Correlation of Blasius: } f = 0.316 \text{Re}^{-0.25}.$$

These are displayed in Figure 5.1 below. As may be observed from the plot the computed values agree reasonably well with measured friction factor and the agreement is better as the Reynolds number increases. This will be more visible at Reynolds number of the order of 10^6 [4]. The $k - \epsilon$ turbulence model appears to work better in the range of Reynolds numbers studied here. For high Reynolds numbers, i.e, fully developed turbulence, the calculated values nearly take after the measured value. Hence most of the results presented here are computed using $k - \epsilon$ turbulence model.

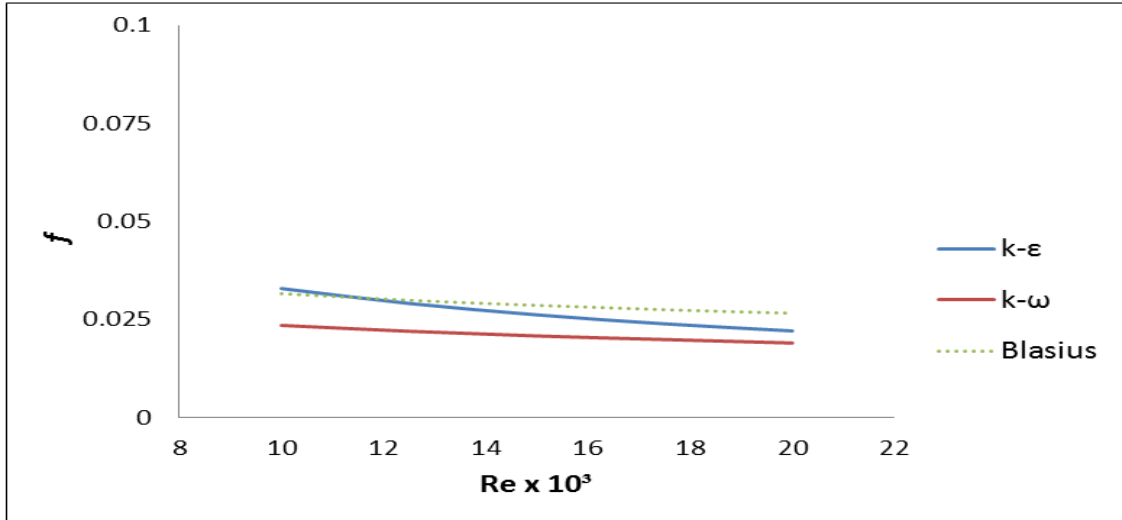


Figure 5.1: Computed friction factors for Smooth pipe

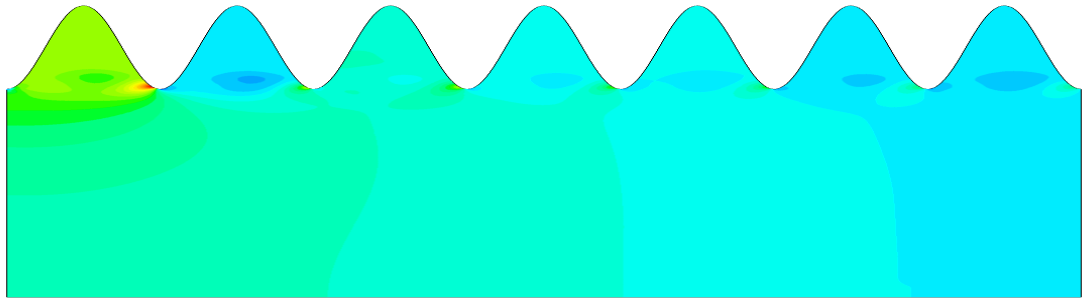
5.4 Results for Corrugated Wall

This section carries the numerical results obtained for the three geometric configurations considered and the smooth pipe, and in addition appropriate tangible interpretations identified with the observed turbulent flow pattern.

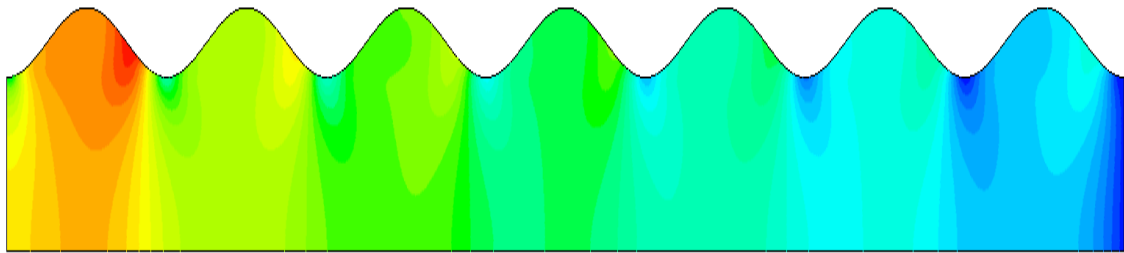
5.4.1 Flow Pattern

5.4.1.1 Total Pressure Contours

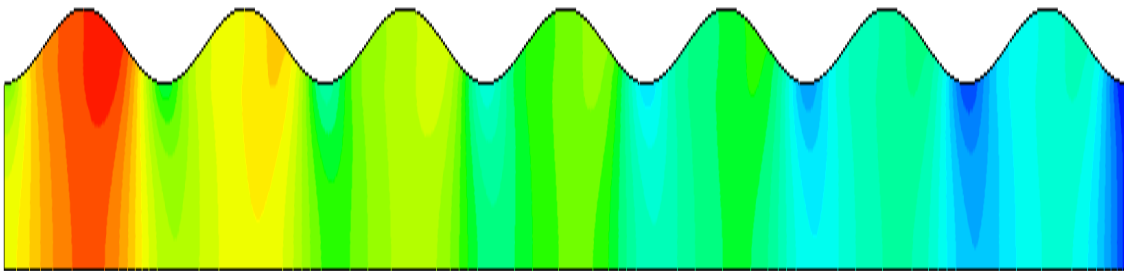
Usually, works including turbulent flow past surfaces with discrete roughness undertake, directly or not, to relate estimated changes in the flows parameters with the flows physical pattern seen. Figures 5.2, 5.3 and 5.4 show total pressure patterns throughout the cross section of the pipe. The figures show that the pressure begins to be inconsistent past the cross section of the pipe, from relatively high pressure to low pressure. Total pressure increases initially and decreases as the flow settles down. Also, it can be observed from these figures that the total pressure increases with λ/a for all Reynolds numbers studied here.



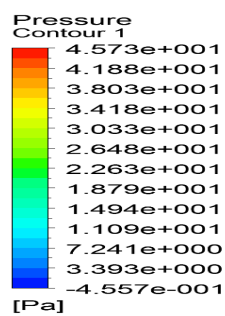
(a) $\lambda/a = 1$ and Re 10000



(b) $\lambda/a = 11$ and Re 10000

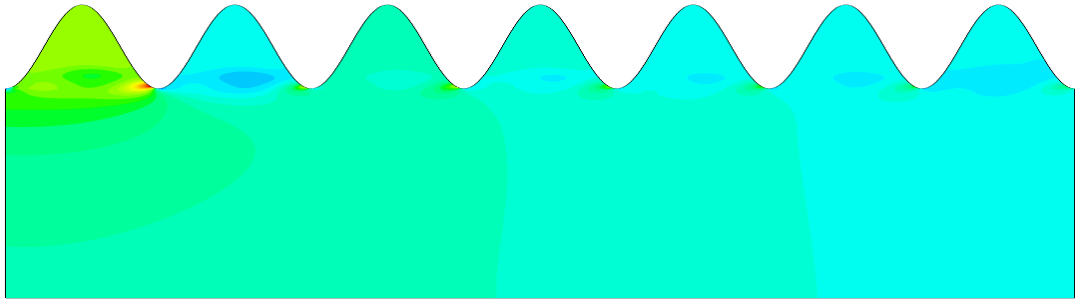


(c) $\lambda/a = 19$ and Re 10000

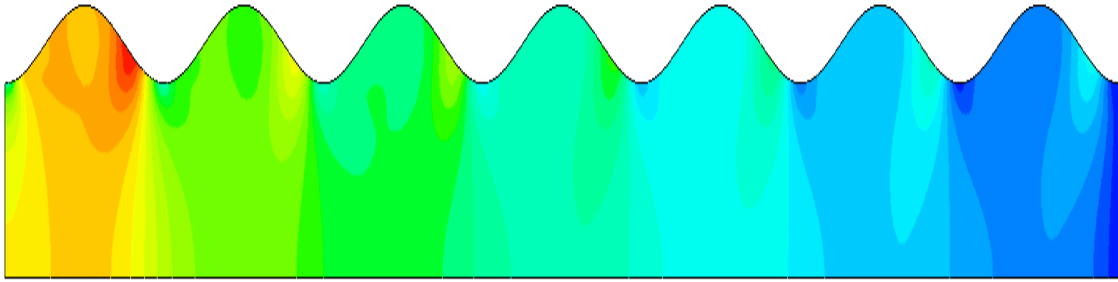


(d) Total Pressure Contour

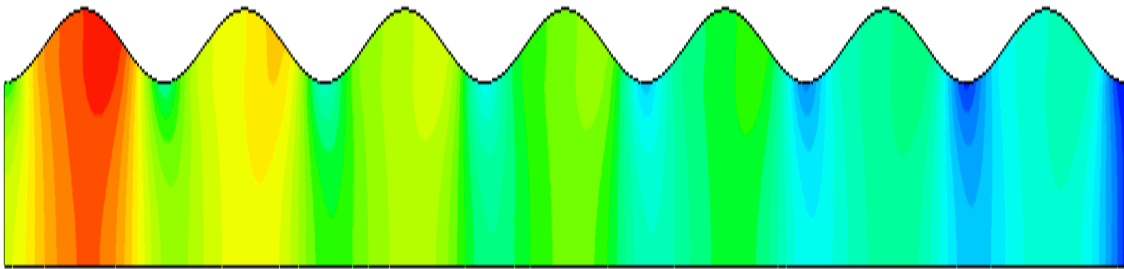
Figure 5.2: Total Pressure Contour at Re 10000



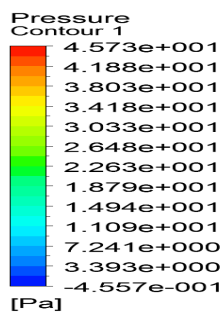
(a) $\lambda/a = 1$ and Re 15000



(b) $\lambda/a = 11$ and Re 15000

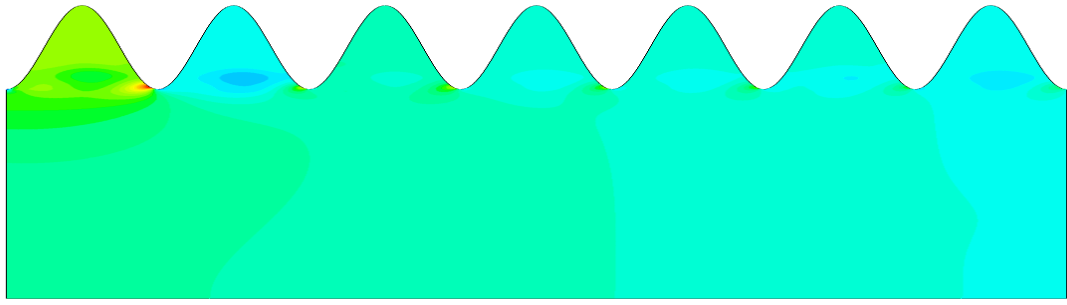


(c) $\lambda/a = 19$ and Re 15000

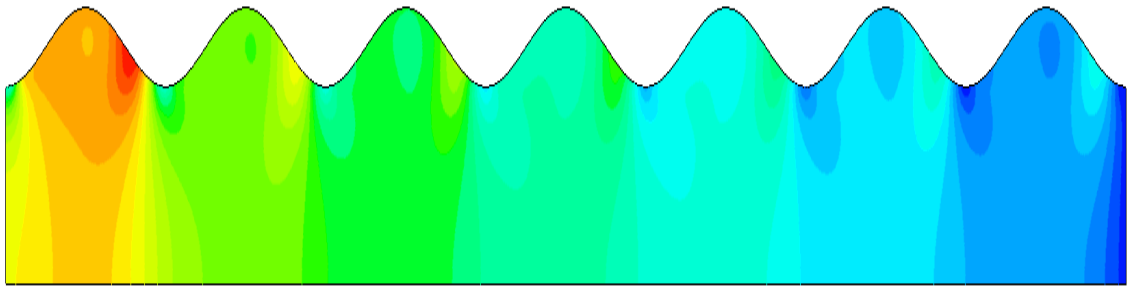


(d) Total Pressure Contour

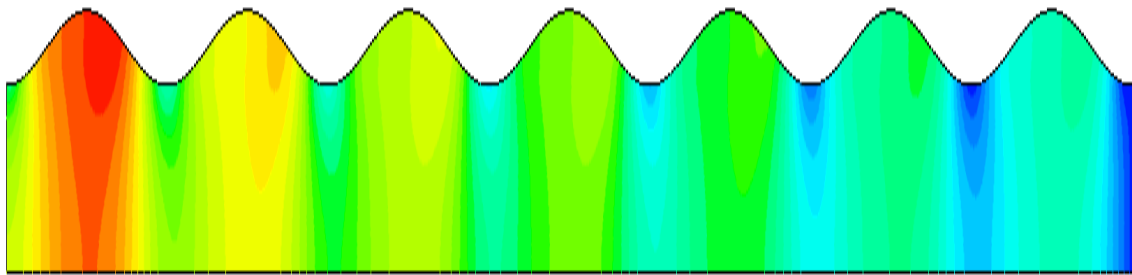
Figure 5.3: Total pressure contour at Re 15000



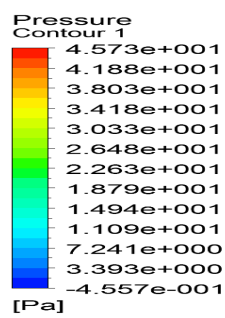
(a) $\lambda/a = 1$ and Re 20000



(b) $\lambda/a = 11$ and Re 20000



(c) $\lambda/a = 19$ and Re 20000



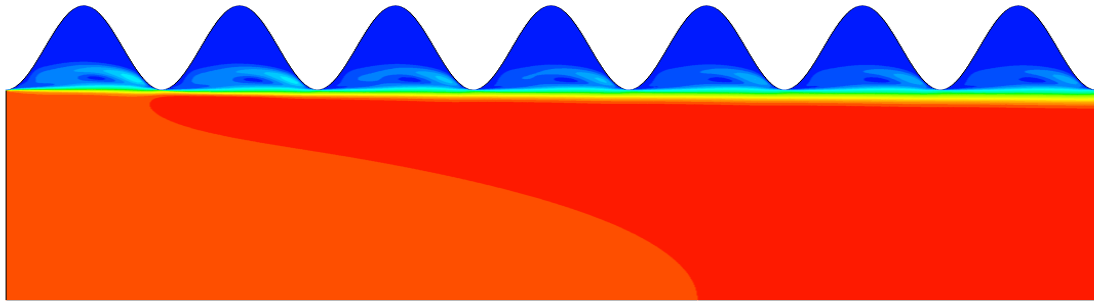
(d) Total Pressure Contour

Figure 5.4: Total pressure contour at Re 20000

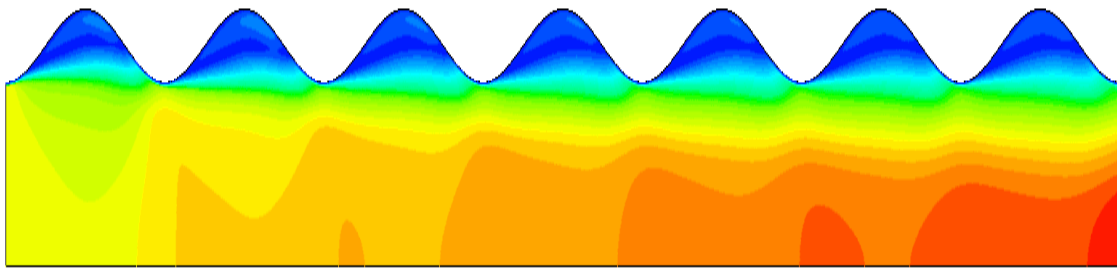
5.4.1.2 Velocity Fields

Flow fields are expressed by velocity vectors and contours. Figures 5.5 - 5.10 show the contours of crossstream velocity magnitude at the cross section of the corrugated pipe. Figures 5.5 - 5.7 show the velocity contours and Figures 5.8 - 5.10 show the velocity vectors across all geometries. From the figures 5.5 - 5.7 it can be seen that there is a recirculation region inside the corrugation (represented by blue colour) and the size of the recirculation reduces as λ/a is increased. As λ/a increases the more fluid is able to flow into the corrugation reducing the size of the recirculation region inside the groove.

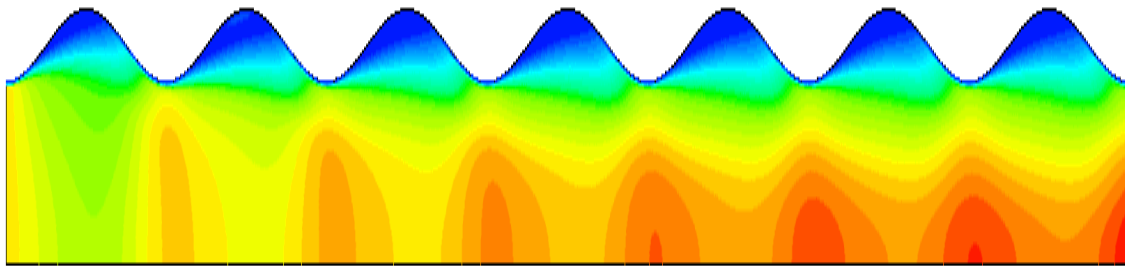
Figures 5.8 - 5.10 show the velocity vectors inside the computational domain. These figures show anti-clockwise vortex inside the groove. Initially symmetric vortex becomes increasingly asymmetric as λ/a increases. Also, it can be seen that the vortex, which is at the interface between the fluid in the groove and fluid outside moves close to of the groove as λ/a increases as more fluid is able to entrain into the groove. This happen in the range of Re studied here.



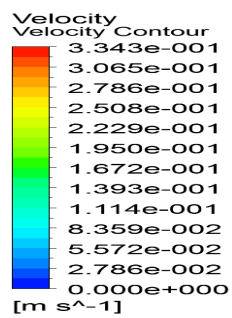
(a) $\lambda/a = 1$ and $Re = 10000$



(b) $\lambda/a = 11$ and $Re = 10000$

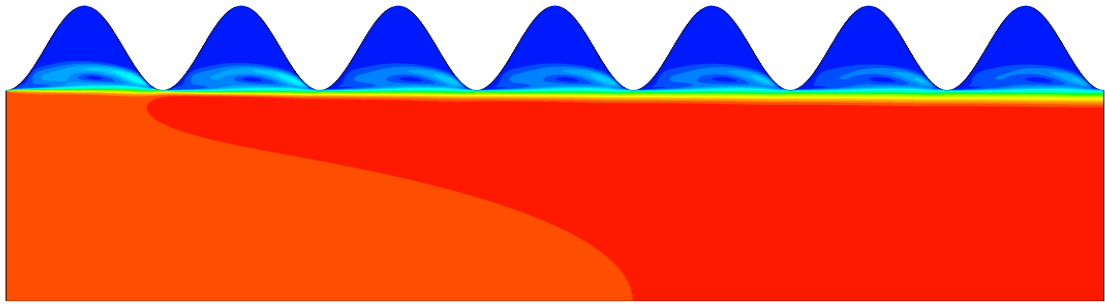


(c) $\lambda/a = 19$ and $Re = 10000$

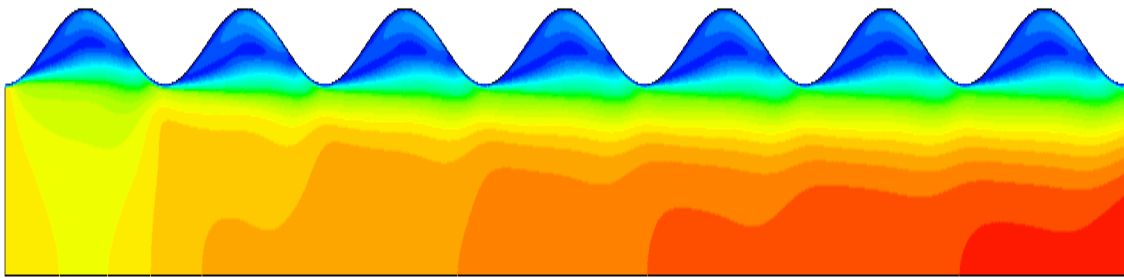


(d) Velocity Contour

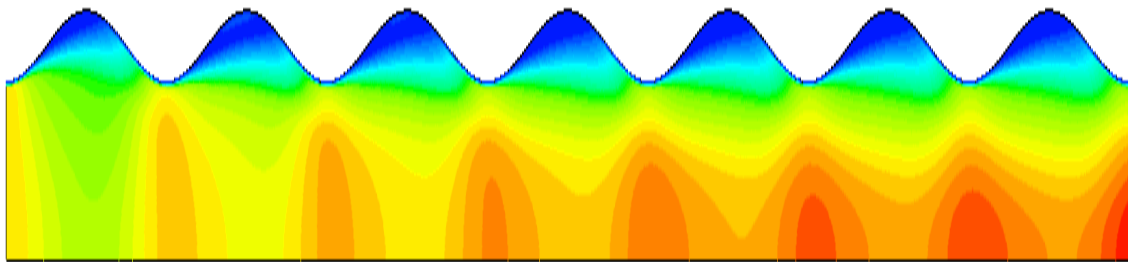
Figure 5.5: Velocity Contour at 10000



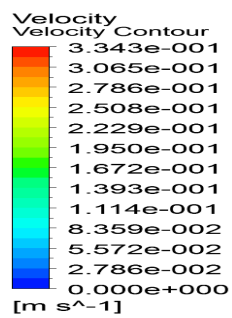
(a) $\lambda/a = 1$ and Re 15000



(b) $\lambda/a = 11$ and Re 15000

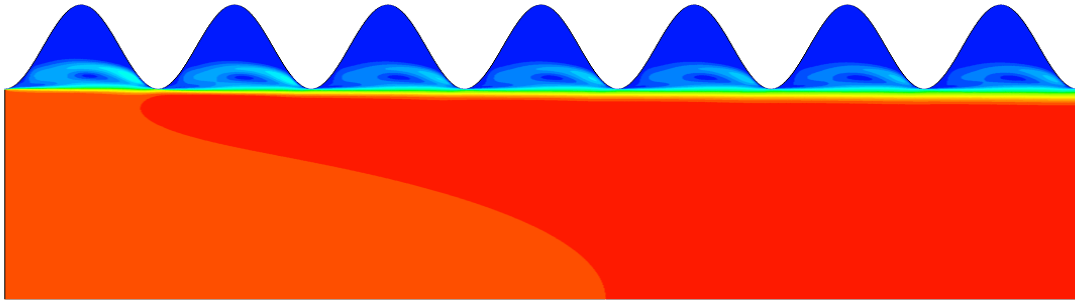


(c) $\lambda/a = 19$ and Re 15000

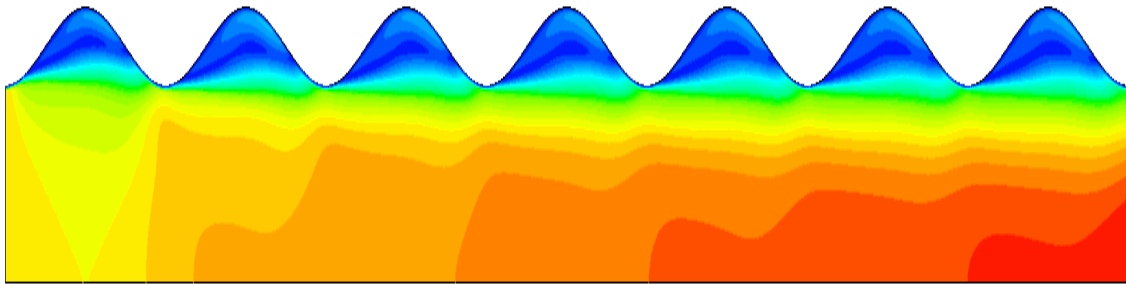


(d) Velocity Contour

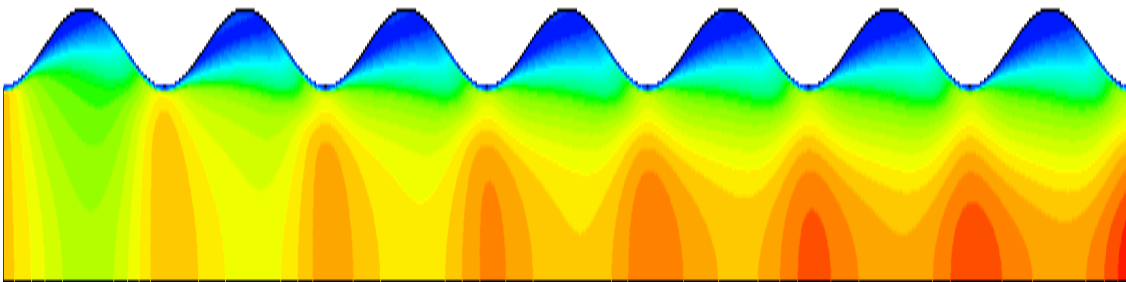
Figure 5.6: Velocity Contour at 15000



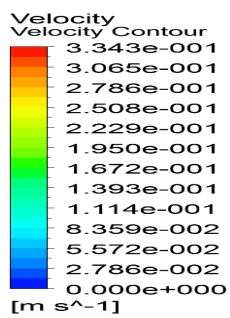
(a) $\lambda/a = 1$ and Re 20000



(b) $\lambda/a = 11$ and Re 20000

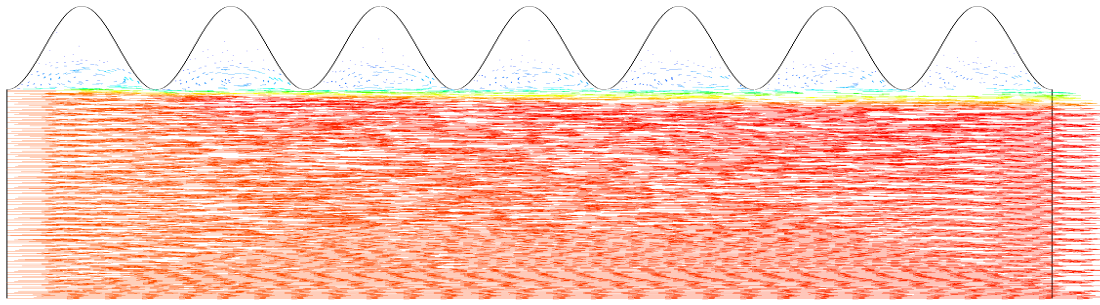


(c) $\lambda/a = 19$ and Re 20000

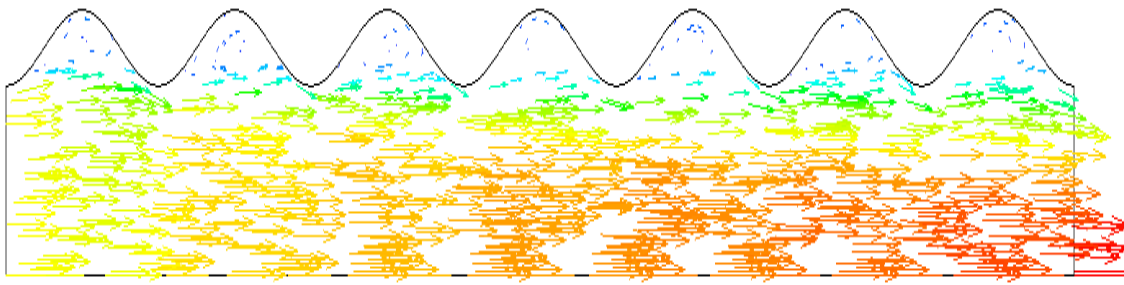


(d) Velocity Contour

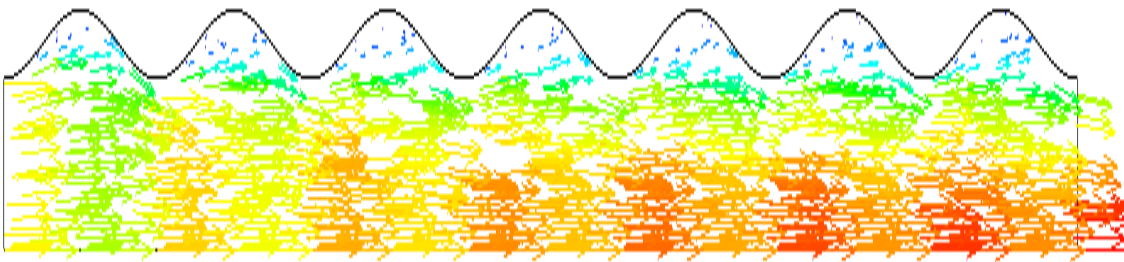
Figure 5.7: Velocity Contour at 20000



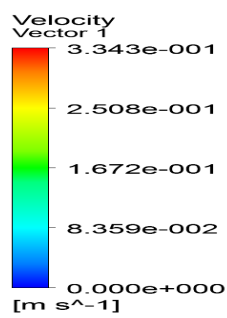
(a) $\lambda/a = 1$ and $Re = 10000$



(b) $\lambda/a = 11$ and $Re = 10000$

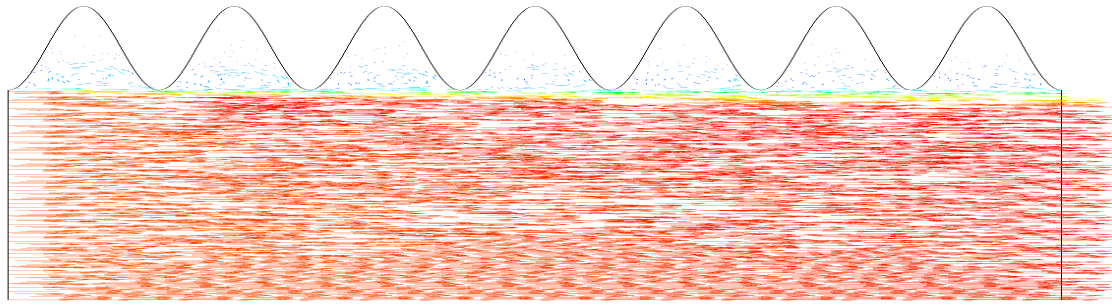


(c) $\lambda/a = 19$ and $Re = 10000$

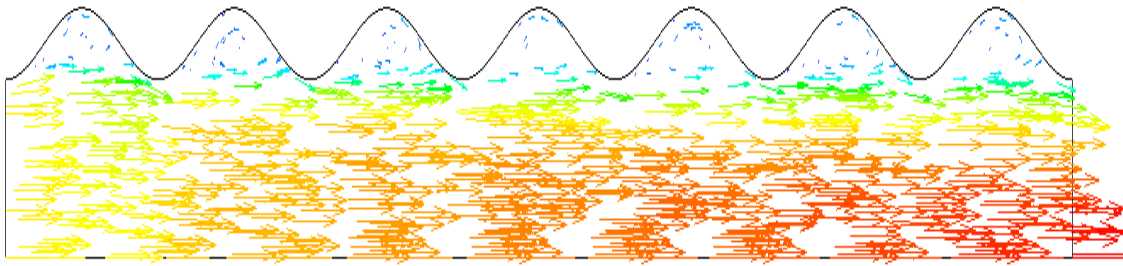


(d) Velocity Vector

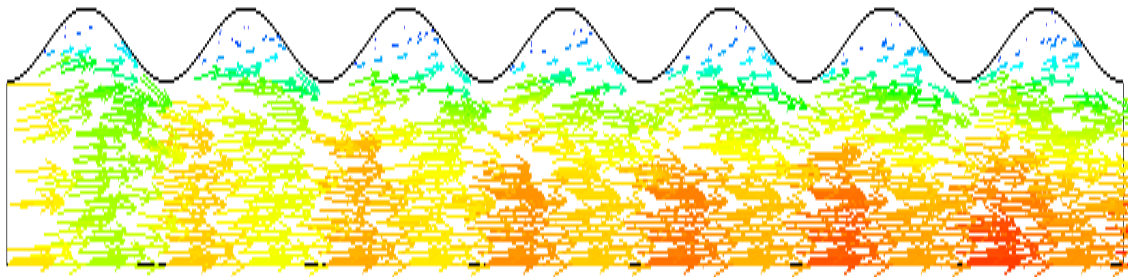
Figure 5.8: Velocity Vectors at $Re = 10000$



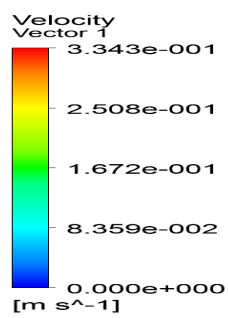
(a) $\lambda/a = 1$ and $\text{Re } 15000$



(b) $\lambda/a = 11$ and $\text{Re } 15000$

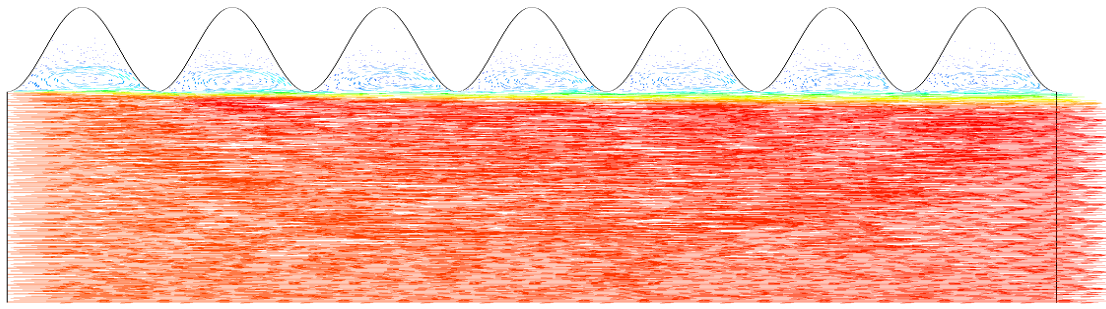


(c) $\lambda/a = 19$ and $\text{Re } 15000$

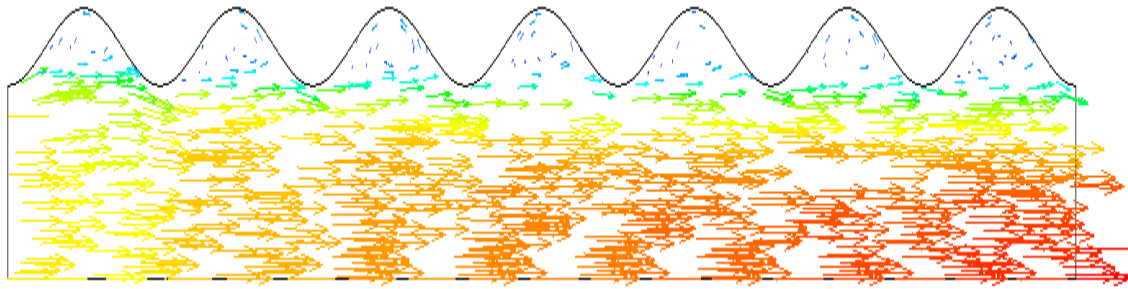


(d) Velocity Vector

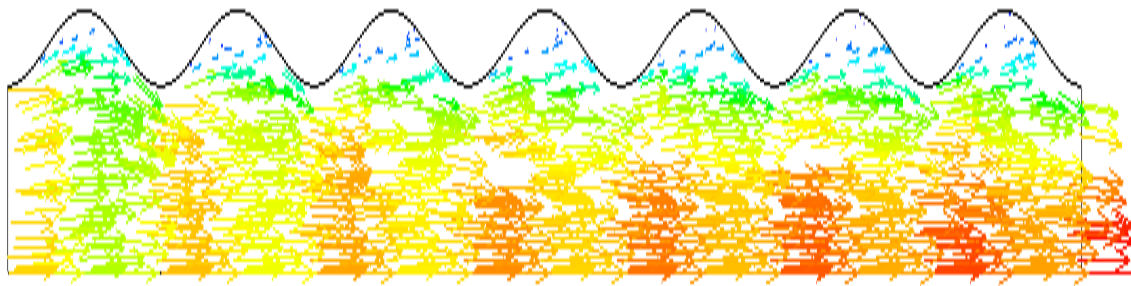
Figure 5.9: Velocity Vectors at $\text{Re } 15000$



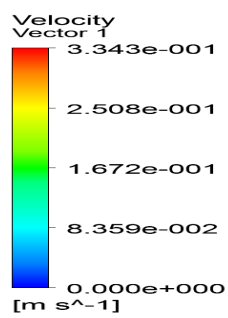
(a) $\lambda/a = 1$ and $Re\ 20000$



(b) $\lambda/a = 11$ and $Re\ 20000$



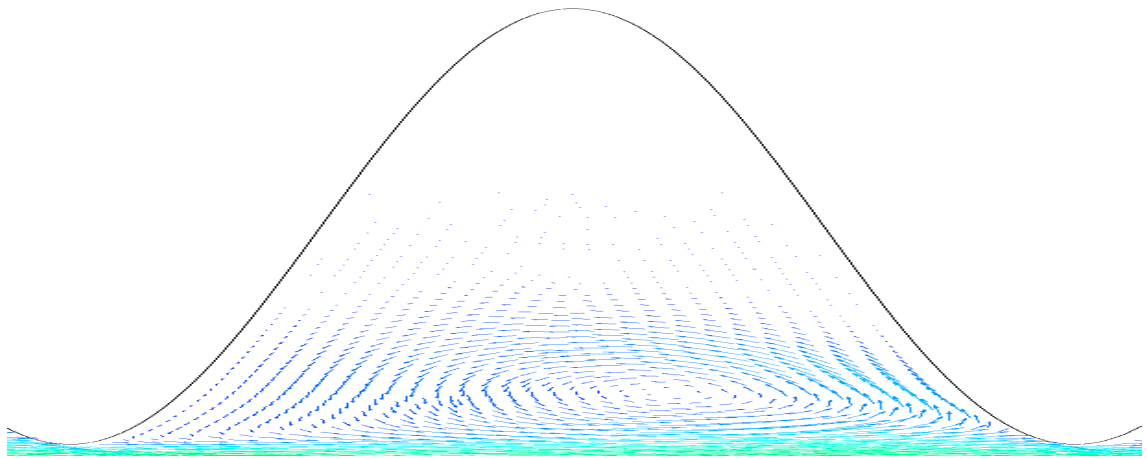
(c) $\lambda/a = 19$ and $Re\ 20000$



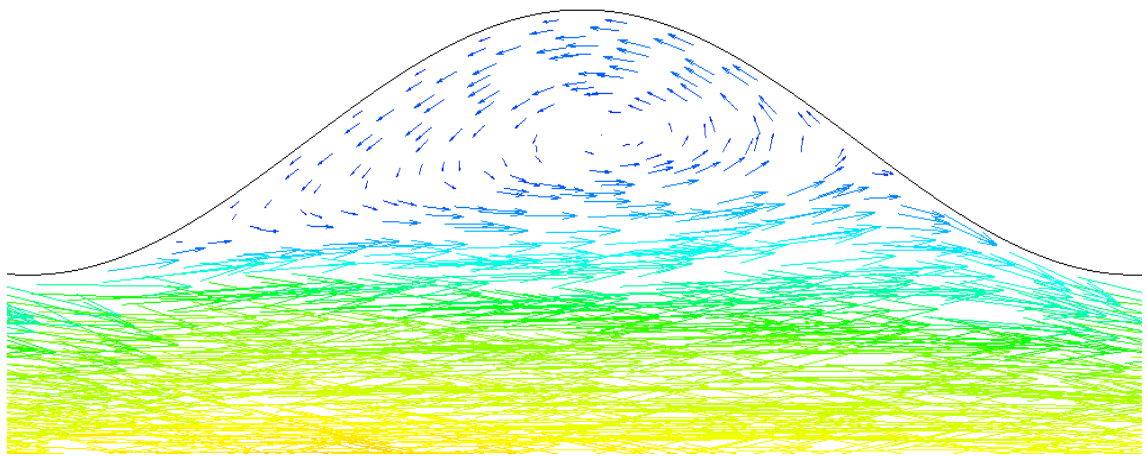
(d) Velocity Vector

Figure 5.10: Velocity Vectors at $Re\ 20000$

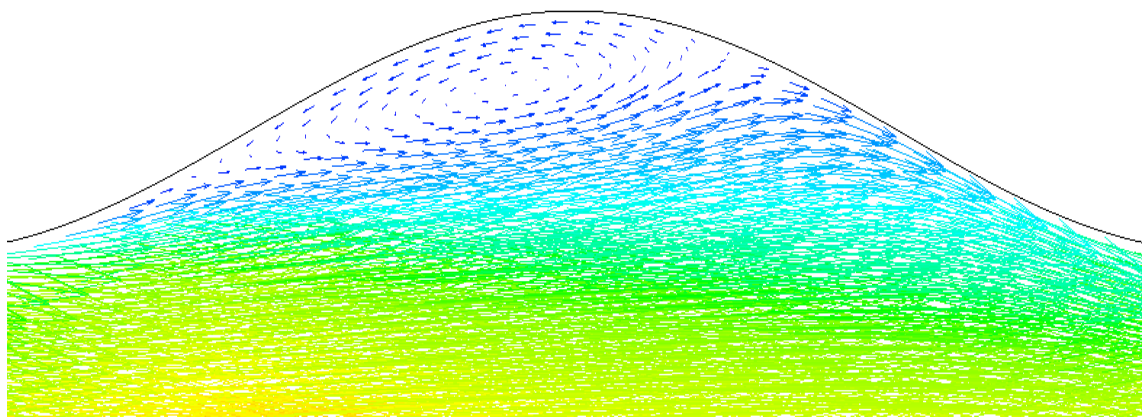
Figures 5.11, 5.12 and 5.13 show the common flow patterns of the enclosed vortex within the corrugation groove seen in the current work. It can be observed, through Figures 5.11, 5.12 and 5.13 that the vortex within the corrugation is significantly affected by the wavy surface. Note that, where the wavy surface is reduced, the enclosed vortex is steady, and completely fills the corrugation, and the interface region between the corrugation and the external flow indicate predominantly axial velocity components. As the wavy surface increases by adjusting the wavelength the vortex center slightly migrates toward the downstream boundary of the corrugation and is compressed, and as a result it loses symmetry as shown by the Figures 5.11a, 5.12a and 5.13a. The current results are consistent with previous numerical results [35–38], where writers have demonstrated that the higher the corrugation wavelength, the earlier the vortex construction in the corrugation region, that the vortex at first structures on the upstream part of the corrugation region, and that as the surface waviness increases, the vortex center slightly dislocates toward downstream boundary of the corrugation region.



(a) $\lambda/a = 1$ and $\text{Re } 10000$

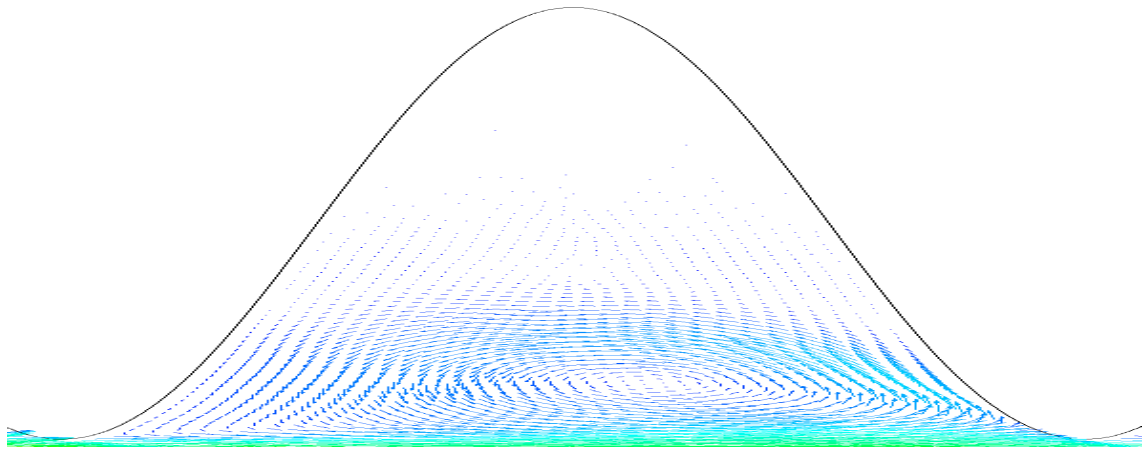


(b) $\lambda/a = 11$ and $\text{Re } 10000$

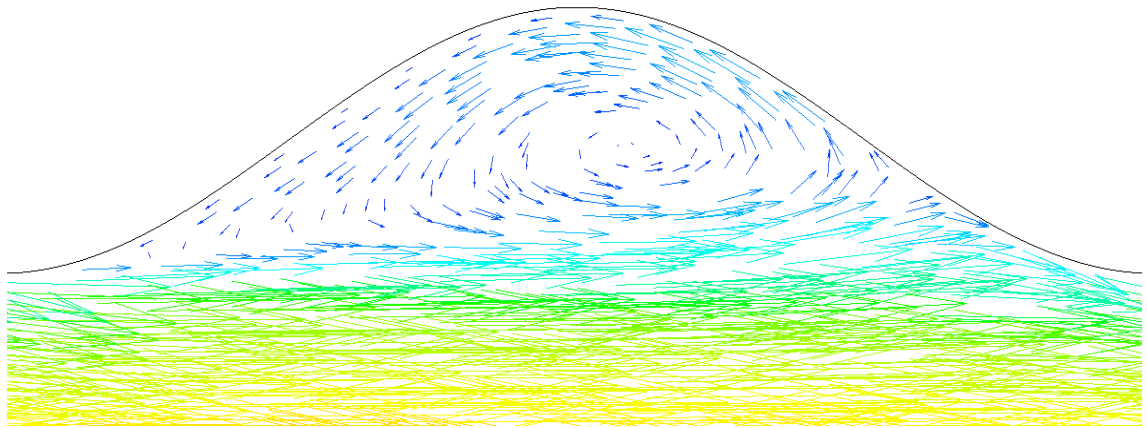


(c) $\lambda/a = 19$ and $\text{Re } 10000$

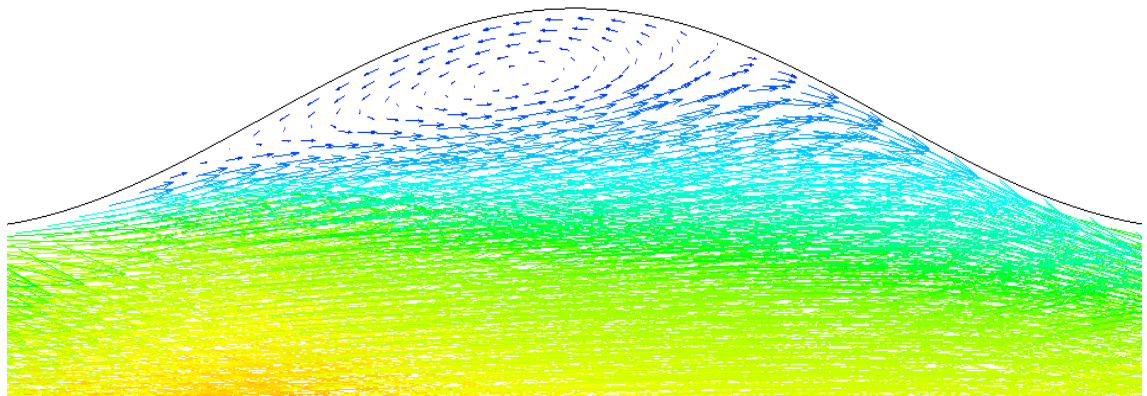
Figure 5.11: Velocity vectors inside the corrugation at $\text{Re } 10000$



(a) $\lambda/a = 1$ and $\text{Re } 15000$

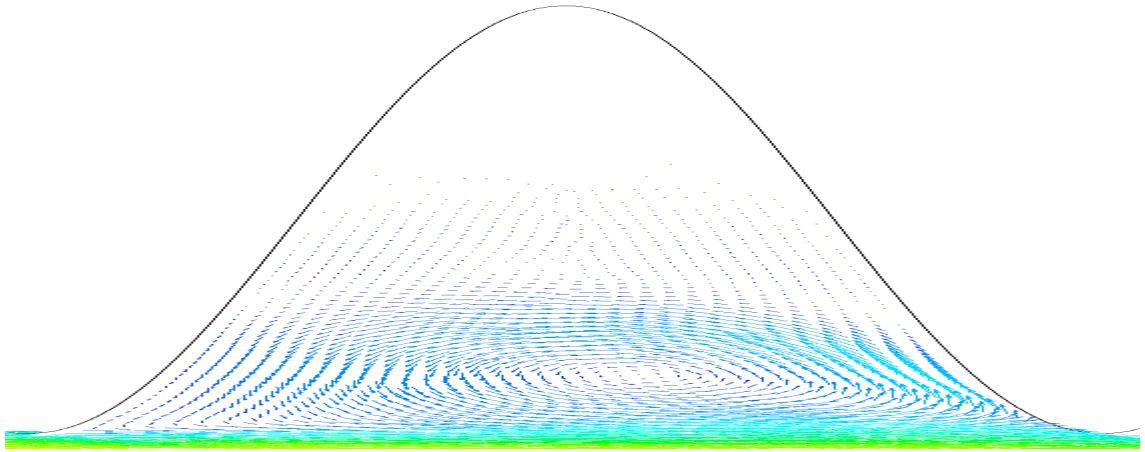


(b) $\lambda/a = 11$ and $\text{Re } 15000$

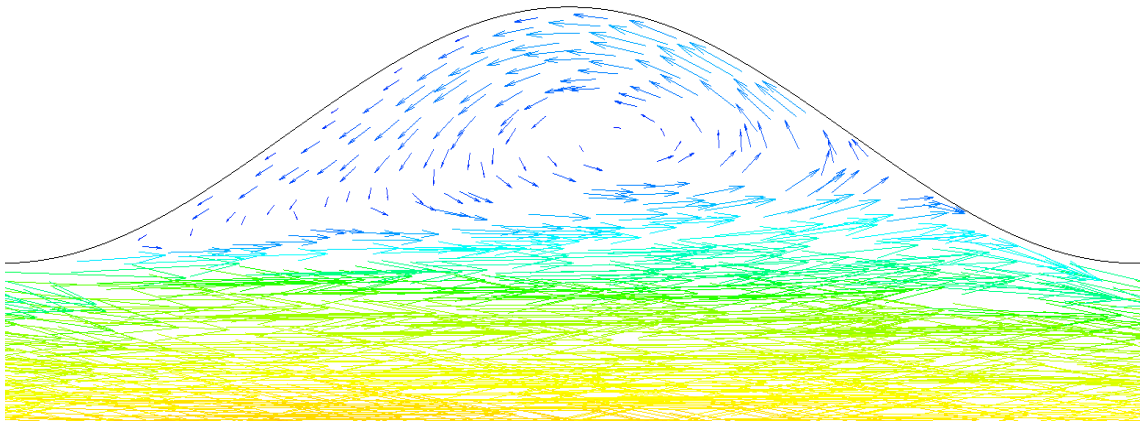


(c) $\lambda/a = 19$ and $\text{Re } 15000$

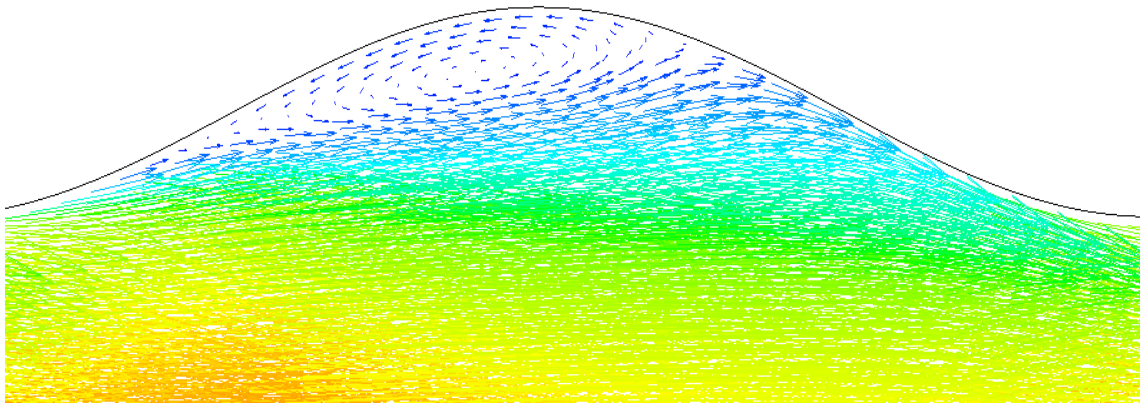
Figure 5.12: Velocity vectors inside the corrugation at $\text{Re } 15000$



(a) $\lambda/a = 1$ and $\text{Re } 20000$



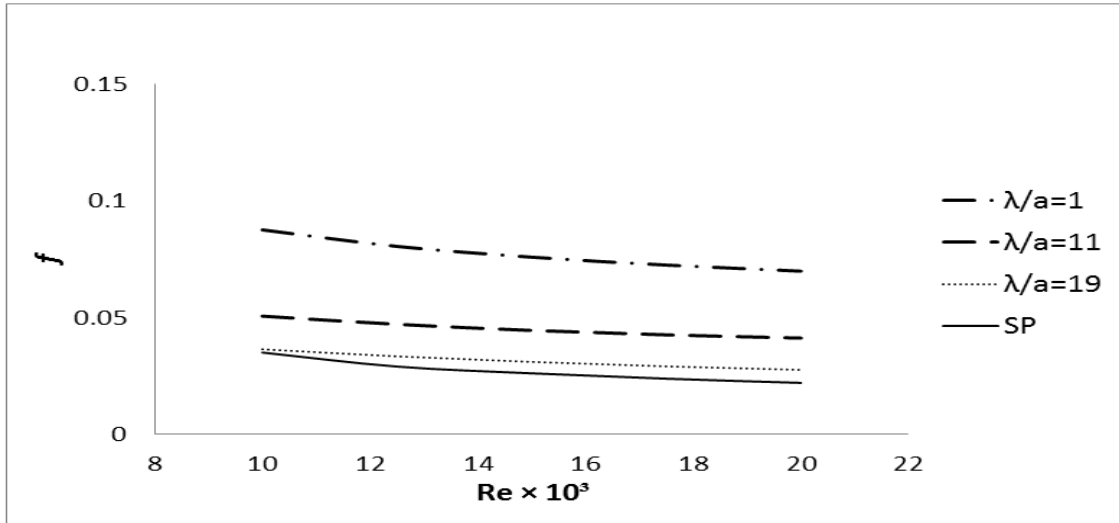
(b) $\lambda/a = 11$ and $\text{Re } 20000$



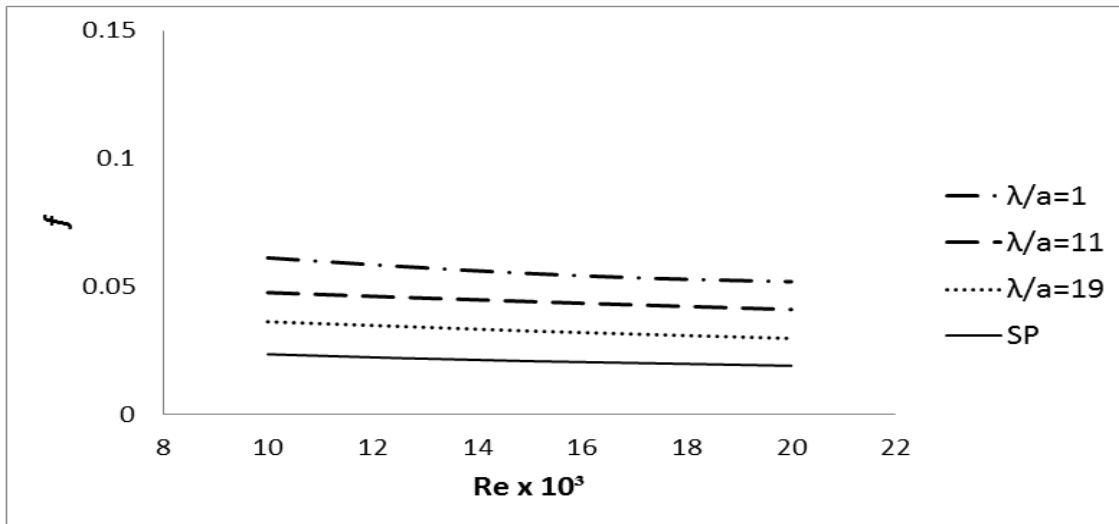
(c) $\lambda/a = 19$ and $\text{Re } 20000$

Figure 5.13: Velocity vectors inside the corrugation at $\text{Re } 20000$

5.4.2 Friction Factor



(a) Using $k - \epsilon$



(b) Using $k - \omega$

Figure 5.14: Numerical friction factor acquired for every geometric configurations and Reynolds numbers projected with smooth pipe.

Figure 5.14 shows the pressure drop on the corrugated pipe obtained in the present work calculated in terms of average friction factor. Figure 5.14 shows the variation of friction factor with Reynolds number at different $\lambda/a = 1, 11, 19$ and Reynolds numbers 10000, 12500, 15000, 17500 and 20000. Figure 5.14a shows the numerical friction factor acquired for all the three geometric configurations and the smooth pipe is plotted as well using $k - \epsilon$ model and Figure 5.14b shows the numerical friction factor acquired for all the three geometric configurations

and the smooth pipe using $k - \omega$ model. Because the simulated Reynolds numbers range is in some way insufficient (10000 – 20000), a logarithmic scale would not be helpful, and differences can be all the more plainly found in a standard conventional scale as demonstrated.

From Figure 5.14, we can see that, large surface waviness demonstrates high frictional loss at a similar Reynolds number. In other words, for a given Reynolds number, the friction factor increases through $\lambda/a = 19$ to 1, signifying that the friction factor decreases as the wavelength increases. Also, from Figure 5.14, it can be observed that for a given Reynolds number the friction factor decrease with increase of λ/a . At the point when the Reynolds number increases, the momentum transfer between the corrugation and the external flow is likewise increased, and this impact is increased when the interface expands. Hence, it could be anticipated that deviations would have a tendency to be greater both for higher Reynolds numbers and for higher corrugation lengths [29, 30]. Consider that, for Reynolds numbers more than 20000, the numerical results acquired show great concurrence with experimental data of Moody's diagram and Nikuradse's diagram [3, 4]. However significant deviations are observed in this range of Reynolds numbers in any case Morales et al. [29], announced some possible errors in this low range, as a result of an equipment limiting range for low mass flow rate.

Again, numerical results for corrugated pipes in comparison to the Blasius smooth pipe correlation demonstrates that the friction factor increases compared to smooth pipes. This increment is more notable for higher Reynolds numbers. Actually, numerical results tend to the smooth pipe correlation for low Reynolds numbers, which is in agreement with one of the typical conclusions of Moody's [4] work, who established that for low Reynolds numbers the friction factor is fundamentally a function of the Reynolds number only.

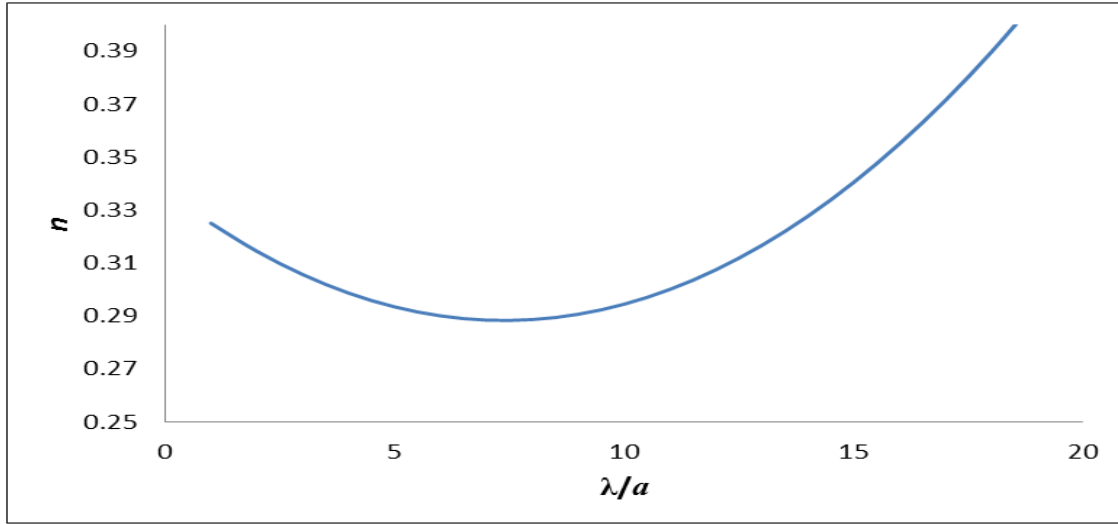


Figure 5.15: Computed friction factors for Smooth pipe

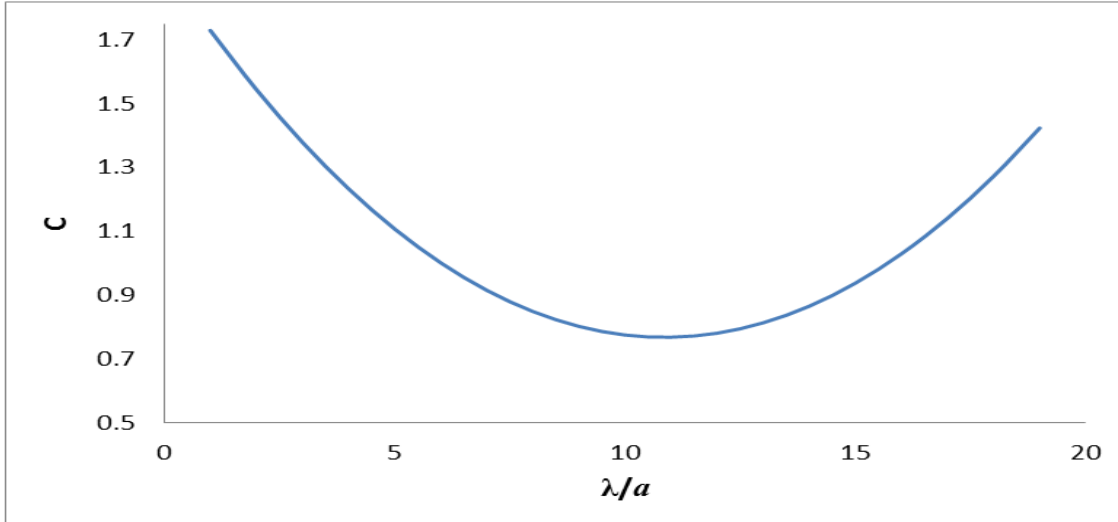


Figure 5.16: Computed friction factors for Smooth pipe

The difference of friction factor with Reynolds number allows to form a correlation between friction factor, Reynolds number and geometric parameter (λ/a). The projected correlation is in the form:

$$f = \frac{C}{\text{Re}^n}. \quad (5.1)$$

Exponent (n) and correlation constant (C) are computed from the slope and intersection of the curves of Figure 5.14a respectively at various geometric parameter (λ/a). The values of ' n ' and ' C ' are then plotted against λ/a as shown in Figure 5.15 and 5.16 respectively. The equation of best-fit curves of Figure 5.15

and 5.16 follows two polynomials of order 2. These are:

$$n = 0.0009(\lambda/a)^2 - 0.0133(\lambda/a) + 0.3375, \quad (5.2)$$

$$C = 0.0099(\lambda/a)^2 - 0.215(\lambda/a) + 1.9351. \quad (5.3)$$

The equations are valid for $1.0 \leq \lambda/a \leq 20.0$, $10000 \leq Re \leq 20000$ and show great agreement with computed data inside $\pm 3\%$ accuracy.

5.5 Performance of Turbulence Models

As stated in the earlier chapters, the computation have been carried out using two turbulence models, namely, $k - \epsilon$ and $k - \omega$ turbulence models. These turbulence models have been described in chapter 3.

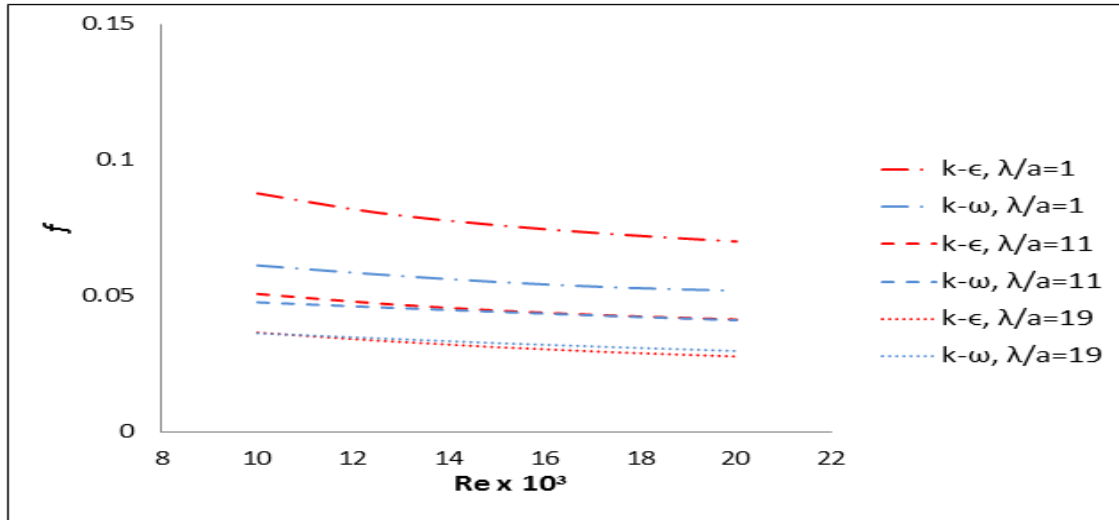


Figure 5.17: Computed friction factor for corrugated pipe using $k - \epsilon$ and $k - \omega$ turbulence models

From these studies, it can be seen that there is no significant difference in the performance of the two turbulence models as shown by Figure 5.17 and Table 5.3

Re	$\lambda/a = 1$	$\lambda/a = 11$	$\lambda/a = 19$
10 000	0.0877	0.0507	0.0365
12 500	0.0806	0.0472	0.0335
15 000	0.0760	0.0446	0.0311
17 500	0.0726	0.0427	0.0292
20 000	0.0700	0.0413	0.0277

(a) $k - \epsilon$ turbulence model

Re	$\lambda/a = 1$	$\lambda/a = 11$	$\lambda/a = 19$
10 000	0.0612	0.0476	0.0362
12 500	0.0579	0.0458	0.0344
15 000	0.0551	0.0441	0.0326
17 500	0.0531	0.0425	0.0311
20 000	0.0519	0.0410	0.0297

(b) $k - \omega$ turbulence model

Table 5.3: Friction factor with $k - \epsilon$ and $k - \omega$ turbulence models

Chapter 6

Conclusion

6.1 Conclusion

Turbulent flow through corrugated pipes has been studied using numerical fluid mechanics. Smooth pipe results have been confirmed with existing available results to validate our methodology. $k - \epsilon$ and $k - \omega$ turbulence models have been used to compute the turbulent flow inside the pipes. The results acquired from the $k - \epsilon$ turbulence model were nearer to the experimental calculations. Hence the $k - \epsilon$ turbulence model was preferred for the simulation of fully developed turbulent flow in longitudinally corrugated pipes. Flow detachment is experienced as fluid moves downstream into diverging portions of the corrugation and reattaching on the converging portion of the corrugation. A vortex is observed inside the corrugation which is influenced extensively by the surface waviness. Where the surface waviness is small, the confined vortex is stable and completely fills the corrugation while increasing the surface waviness the vortex center slightly migrates toward the downstream boundary of the corrugation and is squeezed, and as a result it loses its symmetry. The vortex observed inside the groove is anticlockwise in nature.

Present computational studies have shown that, for this specific geometry, for a given Reynolds number the friction factor decreases with increase of the wavelength. A correlation is proposed for computing friction factor as a function of corrugation surface waviness and Reynolds number.

The performance of two two-equation turbulence models, namely $k - \epsilon$ and $k - \omega$ turbulence models were assessed. They have been proven on smooth and corrugated pipe flows. The pipe flow simulations have proved that the two models perform equally well.

6.2 Recommendations For Further Studies

In this thesis a longitudinally sinusoidal corrugated pipe was studied using $k - \epsilon$ and $k - \omega$ turbulence models over a range of 10 000 - 20 000 Reynolds number. Future research may include other geometries like a transversely sinusoidal corrugated pipe. A wider range of Reynolds number beyond what the current studies used can be considered.

Bibliography

- [1] Mahmud, S., Sadrul Islam A.K.M., Das, P.K., 2001, "Numerical Prediction of Fluid Flow and Heat Transfer in a Wavy Pipe." J. of Thermal Science, Vol. 10, No. 2, pg. 138.
- [2] Pisarenco, M., van der Linden, B., Tijsseling, A., Ory, E., Dam, J., 2011, "Friction Factor Estimation for Turbulent Flows in Corrugated Pipes with Rough Walls," Journal of Offshore mechanics and Arctic Engineering. January 2009
- [3] Nikuradse, J., 1933, "Stromungsgesetz in rauhrenrohren, vDIForschungshefte" 361 (English translation: "Laws of Flow in Rough Pipes."). NACA Technical Memorandum 1292, National Advisory Commission for Aeronautics, Washington, DC, USA.
- [4] Moody, L. F., 1944 "Friction factors for pipe flow" Trans. ASME, 66(8), November, pg. 97-107.
- [5] Lessen, M., and Huang, P., 1976 "Poiseuille Flow in a Pipe With Axially Symmetric Wavy Walls." The physics of Fluids, 19(7), July, pg. 945-950.
- [6] Inaba, T., Ohnishi, H., Miyake, Y., and Murata, S., 1979 "Laminar Flow in a Corrugated Pipe." Bulletin of the JSME, 22(171), September, pg. 1198-1204.
- [7] Blackburn, H. M., Ooi, A., and Chong, M. S., 2007 "The effect of corrugation height on flow in a wavy-walled pipe." In 16th Australasian Fluid Mechanics Conference, A. Editor and B. editor, eds., pg. 559-564.

- [8] Perry, A.E., Schofield, W.H., Joubert, P.N., 1969, "Rough wall turbulent boundary layers", *Journal of Fluid Mechanics*, Vol. 37, pg. 383-413.
- [9] Nishimura, T., Ohori, Y., Kawamura, Y., 1984 "Flow Characteristics in a Channel with Symmetric Wavy Wall For Steady Flow." *J. Chemical Eng. Japan*, pg. 466-471.
- [10] Russ, G., Beer, H., 1997 "Heat Transfer & Flow Field in a Pipe With Sinusoidal Wavy Surface-Numerical Investigation." *Int. J. Heat & Mass Transfer*, 40, pg. 5.
- [11] Yang, L. C., 1997, "Numerical Prediction of Transitional Characteristics of Flow and Heat Transfer in a Corrugated Duct," *ASME J. Heat Transfer*, 119, pg. 62-69.
- [12] Herwig, H., Gloss, D., and Wenterodt, T., 2008 "A New Approach to Understand and Model the Influence of Wall Roughness on Friction Factors for Pipe and Channel Flows," *J. Fluid Mech.*, 613, pg. 35-53.
- [13] Saidi, C., Legay, F., PruentFotch, B., 1987 "Laminar Flow Past a Sinusoidal Cavity." *Int. J. Heat & Mass Transfer*, 30, pg. 4.
- [14] Wang, G., Vanka. S.P., 1995 "Convective Heat Transfer in Periodic Wavy Passages." *Int. J. Heat & Mass Transfer*, 38, pg. 17.
- [15] Russ, G., Beer, H., 1997 "Heat Transfer & Flow Field in a Pipe with Sinusoidal Wavy Surface-II Experimental Investigation." *Int. J. Heat & Mass Transfer*, 40, pg. 5.
- [16] Sparrow, E.M., Hossfeld, L.M., 1984 "Effect of Protruding Edges on Heat Transfer & Pressure Drop in Duct." *Int. J. Heat & Mass Transfer*, 27, pg. 10.
- [17] Yoon, H.S., El-Samni, O.A., Huynh, A.T., Chun, H.H., Kim, H.J., Pham, A.H. and Park, I.R., 2009 "Effect of wave amplitude on turbulent flow in a wavy channel by direct numerical simulation," *Ocean Engineering*, 36, pg. 697-707.

-
- [18] Maass, C. and Schumann, U., 1996 "Direct numerical simulation of separated turbulent flow over a wavy boundary, in Flow Simulation with High Performance Computers," edited by E.H.Hirschel, Notes on Numerical Fluid Mechanics, 52, pg. 227-241.
- [19] Hudson, J.D., Dykhno, L. and Hanratty, T.J., 1996 "Turbulence production in flow over a wavy wall, Experiments in Fluids," 20, pg. 257-265.
- [20] Cherukat, P., Na, Y., and Hanratty, T.J., 1998 "Direct numerical simulations of a fully developed turbulent flow over a wavy wall, Theoretical and Computational Fluid Dynamics," 11, pg. 109-134.
- [21] Chen, J.J., Leung, Y.C., Ko, N.W., 1986, "Drag Reduction in a Longitudinally Grooved Flow Channel", Industrial and Engineering Chemistry Fundamentals, Vol. 25, pg. 741-745.
- [22] Liu, S.K., Klein, S.J., Johnstone, J.P. 1966, Report MD-15, Stanford University: Stanford, CA.
- [23] Kennedy, J.F., Hsu, S.T., Lin J.T., 1973, Journal of the Hydraulics Division, Proceedings of the American Society of Civil Engineers, Vol. 99, pg. 605-616.
- [24] Walsh, M.J., Weinstein, L.M., 1979, "Drag and Heat-Transfer Characteristics of Small Longitudinally Ribbed Surfaces", AIAA Journal, Vol. 17, pg. 770-771.
- [25] Djenidi, L., Anselmet, F., Antonia, R.A., 1994, "LDA measurements in a turbulent boundary layer over a d-type rough wall", Experiments in Fluids, Vol. 16, pg. 323-329.
- [26] Silberman, E., 1970, "Effect of Helix Angle on Flow in Corrugated Pipes", Journal of the Hydraulics Division, Proceedings of the American Society of Civil Engineers, Vol. 96, N HY11, pg. 2253-2263.
- [27] Silberman, E., 1980, "Turbulence in Helically Corrugated Pipe Flow", Journal of the Hydraulics Division, Proceedings of the American Society of Civil Engineers, Vol. 106, N EM4, pg. 699-717.

- [28] Dong, Y., Huixiong, L., Tingkuan, C., 2001, "Pressure Drop, Heat transfer and Performance of Single-Phase Turbulent Flow in Spirally Corrugated Tubes", *Experimental Thermal and Fluid Science*, Vol. 24, pg. 131-138.
- [29] Morales, R.E.M., Franco, A.T., Junqueira, S.L.M, Erthal, R.H., 2007, "Experimental Analysis of Turbulent Flow in Flexible Pipes," Technical Report UTFPR/CENPES-PETOBRA, Curitiba.
- [30] Azevedo, H.S., Morales, R.E.M., Franco, A.T., Junqueira, S.L.M., Erthal, R.H., 2008, "Numerical Simulation of Turbulent Flow in Corrugated Pipe," *Proceedings of the 12th Brazilian Congress of Thermal Engineering and Sciences*, Vol.1, Belo Horizonte, Brazil, pg. 41-44.
- [31] Launder, B.E., and Spalding, D.B., 1972, "Mathematical Models of Turbulence," 169 S.m. Abb. London/New York. Academic Press. Preis geb.
- [32] Wilcox, D.C., 1998, "Turbulence Modeling for CFD," DCW Industries, La Canada, California.
- [33] Jones, P.J., and Whittle, G.E., 1992, "Computational Fluid Dynamics for Building Air Flow Prediction-Current Status and Capabilities." *Building and Environment* 27(3), pg. 321-338
- [34] Blasius, P. R. H., 1913, "Das Aehnlichkeitsgesetz bei Reibungsvorgangen in Flssigkeiten," *Forschungsheft* 131, pg. 1-41.
- [35] Deiber J.A., Schowalter W.R., 1979, "Flow through tubes with sinusoidal axial variations in diameter." *AIChE Journal*, 25(4), pg. 638-645.
- [36] Nishimura. T., Bian, Y.N., Matsumoto, Y., Kunitsugu, K., 2003, "Heat Mass Transfer," 39, pg. 239-248.
- [37] Mahmud S, Islam A.K.M.S., Feroz, C.M., 2003, "Heat Mass Transfer," 39, pg. 387-393.
- [38] Lahbabi, A., Chang, H., 1986, *Chem Eng Sci* 41, pg. 2487-2505.