

151857

ÇUKUROVA UNIVERSITY
INSTITUTE OF NATURAL AND APPLIED SCIENCES

PhD THESIS

Azize AKKOCA

**COMPUTATIONAL MODELING OF TURBULENT HEAT TRANSFER
IN PLATE FIN AND TUBE HEAT EXCHANGERS**

DEPARTMENT OF MECHANICAL ENGINEERING

ADANA, 2004

151857

**ÇUKUROVA ÜNİVERSİTESİ
FEN BİLİMLERİ ENSTİTÜSÜ**

**COMPUTATIONAL MODELING OF TURBULENT
HEAT TRANSFER IN PLATE FIN AND TUBE HEAT
EXCHANGERS**

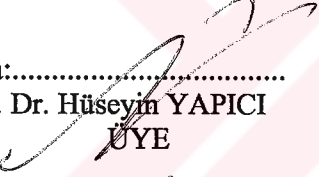
Azize AKKOCA

DOKTORA TEZİ

MAKİNA MÜHENDİSLİĞİ ANABİLİM DALI

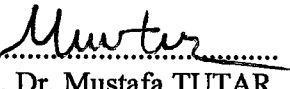
**Bu Tez 09/08/2004 Tarihinde Aşağıdaki Jüri Üyeleri Tarafından
Oybirliği/Oyçokluğu İle Kabul Edilmiştir.**

İmza: 
Prof. Dr. Beşir ŞAHİN
DANIŞMAN

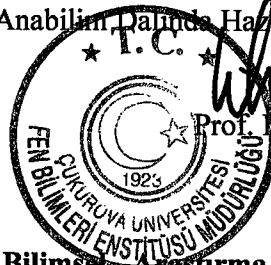
İmza: 
Prof. Dr. Hüseyin YAPICI
ÜYE

İmza: 
Prof. Dr. Orhan BÜYÜKALACA
ÜYE

İmza: 
Yrd. Doç. Dr. Hüseyin AKILLI
ÜYE

İmza: 
Yrd. Doç. Dr. Mustafa TUTAR
ÜYE

**Bu Tez Enstitümüz Makina Mühendisliği Anabilim Dalında Hazırlanmıştır.
Kod No: 8391**



İmza: 
Prof. Dr. Fikri AKDEMİR
Enstitü Müdürü
İmza ve Mühür

**Bu çalışma Çukurova Üniversitesi Bilimsel Araştırma Projeleri Birimi
tarafından desteklenmiştir. Proje No: FBE.2001.D.51**

Not: Bu tezde kullanılan özgün ve başka kaynaktan yapılan bildirişlerin, çizelge, şekil ve fotoğrafların kaynak gösterilmeden kullanımı, 5846 sayılı Fikirve Sanat Eserleri Kanunundaki hükümlere tabidir.

ABSTRACT

PhD THESIS

COMPUTATIONAL MODELING OF TURBULENT HEAT TRANSFER IN PLATE FIN AND TUBE HEAT EXCHANGERS

Azize AKKOCA

**DEPARTMENT OF MECHANICAL ENGINEERING
INSTITUTE OF NATURAL AND APPLIED SCIENCES
UNIVERSITY OF ÇUKUROVA**

**Supervisor : Prof. Dr. Beşir ŞAHİN
Year: 2004, Pages: 230**

**Jury : Prof. Dr. Beşir ŞAHİN
: Prof. Dr. Hüseyin YAPICI
: Prof. Dr. Orhan BÜYÜKALACA
: Assist. Prof. Dr. Hüseyin AKILLI
: Assist. Prof. Dr. Mustafa TUTAR**

Knowledge of thermo-hydraulic fundamentals for plate fin and tube heat exchangers is necessary to successfully carry out thermal design such as heat transfer and pressure drop data. In order to provide an optimum solution of these type of heat exchangers, Computational Fluid Dynamics (CFD) and Computational Heat Transfer (CHT) techniques are applied by employing both laminar and turbulence models. In the first part of the study, the present finite volume based laminar modeling technique is tested and validated against experimental results in terms of heat transfer and pressure drop performance of the heat exchanger under laminar flow conditions. In the second part, in order to investigate the effect of turbulence on the present flow, turbulence model simulations are carried out using both time averaged and space filtered Navier Stokes Equations based turbulence models and the best turbulence model was chosen as Large Eddy Simulation (LES) in comparison with the experimental data on the basis of the fluid flow and heat transfer calculations. In the last part of the study, in order to consider time and space variations of turbulence effects, a dynamic LES method based on Yoshizawa is adapted for the present calculations to improve the capability of the model. The validation of the present numerical model is tested against the experimental data selected from the literature and finally it is verified by using Particle Image Velocimetry (PIV) technique.

Keywords: Computational Fluid Dynamics (CFD), Computational Heat Transfer (CHT), Plate Fin and Tube Heat Exchanger, Large Eddy Simulation (LES), Particle Image Velocimetry (PIV)

ÖZ

DOKTORA TEZİ

KANAT- BORULU ISI DEĞİŞTİRİCİLERDE TÜRBÜLANSLI ISI TRANSFERİNİN BİLGİSAYARLI MODELLENMESİ

Azize AKKOCA

ÇUKUROVA ÜNİVERSİTESİ
FEN BİLİMLERİ ENSTİTÜSÜ
MAKİNA MÜHENDİSLİĞİ ANABİLİM DALI

Danışman : Prof. Dr. Beşir ŞAHİN
Yıl: 2004, Sayfa: 230

Jüri : Prof. Dr. Beşir ŞAHİN
: Prof. Dr. Hüseyin YAPICI
: Prof. Dr. Orhan BÜYÜKALACA
: Yrd. Doç. Dr. Hüseyin AKILLI
: Yrd. Doç. Dr. Mustafa TUTAR

Bilindiği gibi, kanat-borulu kompakt bir ısı değıştircisinin termal-hidrolik özelliklerinin bilinmesi, bu tip ısı değıştircilerin ısı transferi ve basınç farkı gibi tasarım parametrelerinin belirlenmesi için gereklidir. Bu çalışmada Hesaplı Akışkanlar Dinamiği (HAD) ve Hesaplı Isı Transferi (HIT) yöntemleri kullanılarak, kanat-borulu kompakt bir ısı değıştircisinin performansını etkileyen optimum geometrik parametrelerin büyüklükleri, çalışmanın ilk aşamalarında laminar model kullanılarak belirlenmiş; daha sonraki aşamalarda türbülansın ısı değıştircideki lokal akım ve ilgili ısı transferi üzerindeki etkilerini araştırmak amacıyla türbülans modelleri kullanılmış ve en iyi performansı gösteren türbülans modeli Büyük Eddy Simülasyonu (BES) olarak belirlenmiştir. Çalışmanın son aşamasında, daha doğru sayısal çözümleme yapabilmek için, türbülansın zaman ve koordinat etkileri de göz önüne alınarak, dinamik bir BES metodu önerilmiş (Yoshizawa modeli) ve bu dinamik model, daha önce geometrik özellikleri belirlenen optimum ısı değıştirciye göre uyarlanmıştır. Çalışmanın bir çok bölümlerinde literatürden seçilen deneysel çalışmalarla karşılaştırılan nümerik model, Parçacık Görüntülemeli Hız Ölçme (PIV) tekniği kullanılarak, deneysel olarak doğrulanmıştır.

Anahtar Kelimeler: Hesaplı Akışkanlar Dinamiği (HAD), Hesaplı Isı Transferi (HIT), Kanat- Borulu Isı Değıştirci, Büyük Eddy Simülasyonu (BES), Parçacık Görüntülemeli Hız Ölçüm Tekniği (PIV)

ACKNOWLEDGEMENT

First of all, I would like to express my deepest gratitude to my thesis supervisor, Prof. Dr. Beşir ŞAHİN, for his guidance, continual encouragement, endless support, invaluable understanding and patience throughout the study. I also owe special thanks to him for providing me with the opportunity to attend my study. Without him, I would never have completed my study.

I would like to express my most sincere gratitude to Dr. Mustafa TUTAR, who introduced me to the subject of Computational Fluid Dynamics. His constructive comments, suggestions, criticism, invaluable remarks, and expertise have greatly contributed to this study especially when he was the second supervisor at the beginning of this study. I would like to thank him for his guidance and helping me to develop new insights in my study.

I wish to extend my gratitude to Dr. Hüseyin AKILLI for his expert comments on the improvement of postprocessing, invaluable suggestions on boundary conditions, and conducting experiments for the present investigation.

A part of the study has been conducted at Department of Mechanical Engineering at Mersin University by the research grant obtained from TUBITAK under the project No: MISAG-147. So, I would like to acknowledge Scientific and Technical Research Council of Turkey (TUBITAK). I also would like to thank to the Computational Fluid Dynamics Group members of Mechanical Engineering Department of Mersin University for allowing me construct all the geometries and meshes used in this study by using their computers and softwares and for use their library and documentation facilities.

I owe thanks to all my friends of the Fluid Mechanics Research Group of Mechanical Engineering Department of Çukurova University, especially to Nurhan Adil ÖZTÜRK, Cuma KARAKUŞ, Atakan AKAR, Coşkun ÖZALP, and Cahit GÜRLEK for helping me with the computer and conducting the experiments.

Finally, my most sincere and heartfelt appreciation go to my son, Evren Cem, who has always reminded me of life and joy and complained that I have been always studying instead of playing with him.

CONTENTS

PAGE

ABSTRACT.....	I
ÖZ.....	II
ACKNOWLEDGEMENT.....	III
CONTENTS.....	IV
LIST OF FIGURES.....	VII
LIST OF TABLES.....	XIX
NOMENCLATURE.....	XX
1.INTRODUCTION.....	1
2. BACKGROUND AND LITERATURE SURVEY.....	9
2.1. Geometrical and Flow Characteristics of Plate Fin and Tube Heat Exchangers.....	10
2.1.1. Definition of Horseshoe Vortex.....	17
2.1.2. Boundary Layer Separation and Flow Around a Circular Cylinder.....	17
2.2. Fluid flow Topology.....	12
2.3. Literature Survey for Plate Fin and Tube Heat Exchangers.....	12
2.3.1. Experimental Studies.....	17
2.3.1. Numerical Studies.....	17
3. OBJECTIVES AND OUTLINE OF PRESENT INVESTIGATION.....	55
3.1. Overview of Unresolved Issues.....	55
3.2. Outline of Dissertation.....	57
4. MATERIALS AND METHODS.....	33
4.1. Governing Equations for Fluid Flow and Heat Transfer.....	34
4.2. Reynolds Averaged Navier Stokes Equations of Motion.....	35
4.3. Overview of Turbulence Modeling.....	37
4.4. Turbulence Models.....	38
4.4.1. Time Averaged Turbulence Models.....	38
4.4.1.1. Zero Equation Model.....	38
4.4.1.2. One Equation Model.....	39

CONTENTS

PAGE

ABSTRACT.....	I
ÖZ.....	II
ACKNOWLEDGEMENT.....	III
CONTENTS.....	IV
LIST OF FIGURES.....	VII
LIST OF TABLES.....	XV
NOMENCLATURE.....	XVI
1.INTRODUCTION.....	1
2. BACKGROUND AND LITERATURE SURVEY.....	9
2.1. Geometrical and Flow Characteristics of Plate Fin and Tube Heat Exchangers.....	10
2.1.1. Definition of Horseshoe Vortex.....	17
2.1.2. Boundary Layer Separation and Flow Around a Circular Cylinder.....	17
2.2. Fluid flow Topology.....	12
2.3. Literature Survey for Plate Fin and Tube Heat Exchangers.....	12
2.3.1. Experimental Studies.....	17
2.3.1. Numerical Studies.....	17
3. OBJECTIVES AND OUTLINE OF PRESENT INVESTIGATION.....	55
3.1. Overview of Unresolved Issues.....	55
3.2. Outline of Dissertation.....	57
4. MATERIALS AND METHODS.....	33
4.1. Governing Equations for Fluid Flow and Heat Transfer.....	34
4.2. Reynolds Averaged Navier Stokes Equations of Motion.....	35
4.3. Overview of Turbulence Modeling.....	37
4.4. Turbulence Models.....	38
4.4.1. Time Averaged Turbulence Models.....	38
4.4.1.1. Zero Equation Model.....	38
4.4.1.2. One Equation Model.....	39

CONTENTS

PAGE

4.4.1.3. Two Equation Models.....	40
4.4.1.4. Reynolds Stress Model.....	44
4.4.1.5. Algebraic Stress Model.....	45
4.4.2. Space Averaged (Filtered) Turbulence Models.....	45
4.4.2.1. Smagorinsky- Lily Sub- Grid Scale Model.....	47
4.4.2.2. RNG- Based Sub- Grid Scale Model.....	48
4.4.2.3. Dynamical LES Model: The Yoshizawa TSDIA Model (1984).....	49
4.4.3. Direct Numerical Simulation.....	51
4.5. Near Wall Treatment.....	51
4.6. Comparison of Different Turbulence Models.....	54
4.7. Overview of Computational Fluid Dynamics and Computational Heat Transfer.....	55
4.8. Numerical Method.....	56
4.9. Heat Exchanger Model.....	57
4.10. Computational Mesh.....	58
4.11. Boundary Conditions.....	65
4.11.1. Boundary Conditions for Steady State Simulations.....	65
4.11.2. Boundary Conditions for Transient Simulations.....	67
4.11.3. Boundary Conditions for Turbulent Simulations.....	68
4.12. Computational Details.....	70
4.12.1. Discretization.....	71
4.12.1.1. Second Order Upwind Scheme.....	72
4.12.1.2. Under Relaxation.....	73
4.12.1.3. Pressure- Velocity Coupling.....	73
4.12.2. Numerical Scheme.....	75
4.12.3. Convergence.....	76
4.13. Experimental Study.....	77
4.13.1. Experimental Set-Up.....	77

CONTENTS

PAGE

4.13.2. Test Section.....	79
5. RESULTS AND DISCUSSION.....	81
5.1. Grid Independency Analysis and Experimental Verification.....	81
5.2. Results of Optimization Studies for Laminar and Steady State Flow.....	85
5.2.1. Effect of Fin Spacing.....	85
5.2.2. Effect of Number of Tube Rows.....	97
5.2.3. Effect of Cylinder Layout (Inlined-Staggered).....	99
5.2.4. Effect of Fin Configuration (Plain-Wavy).....	104
5.2.5. Effect of Wavy Angle.....	104
5.3. Effect of Reynolds Number on the Prediction of Flow Characteristics.....	105
5.4. Effect of Selection of Different Wall Functions on the Heat Transfer and Pressure Drop in the Heat Exchanger Flow Passage.....	117
5.5. Turbulence Model Results.....	121
5.6. Laminar Model Transient Simulation Results.....	140
5.7. Experiment Results.....	161
6. CONCLUSIONS.....	195
7. FUTURE WORK.....	198
REFERENCES.....	199
CURRICULUM VITAE.....	208
APPENDIX A.....	210
APPENDIX B.....	221
APPENDIX C.....	230

LIST OF FIGURES**PAGE**

Figure 1.1.	Classification of heat exchangers (Kuppan, 2000).....	4
Figure 1.2.	Flat or continuous fins on an array of tubes (Rohsenow et al, 1998).....	5
Figure 1.3.	Plain in configurations (Kakac and Liu, 1997) and wavy fin configurations (Wang et al., 1999).....	8
Figure 2.1.	Local scour mechanism at the base of the bridge pier (Sheppard, 2004).....	12
Figure 2.2.	Flow patterns associated with local scour (Miller, 2003).....	13
Figure 2.3.	Sketches of a bridge pier in a river bed (Muzzammil and Ganngadhariah, 2003).....	14
Figure 2.4.	Schematic representation of the area of the horseshoe vortex (Fotherby, 1995).....	15
Figure 2.5.	Singular points (Waclawicz, 2001).....	19
Figure 2.6.	Flow around a surface mounted cubical obstacle (Xing and Baili, 2004).....	20
Figure 4.1.	Subdivisions of near wall region (FLUENT, 1998).....	52
Figure 4.2.	Near- Wall Treatments (FLUENT, 1998).....	53
Figure 4.3.	Computational domain for the heat exchanger model.....	57
Figure 4.4.	Detail of mesh on the fin surface.....	59
Figure 4.5.	Detail of mesh on the tube surface.....	60
Figure 4.6.	Global mesh of the computational domain for the single cylinder configuration.....	61
Figure 4.7.	Global mesh of the computational domain for the two cylinder of inlined layout.....	61
Figure 4.8.	Global mesh of the computational domain for the three cylinder of inlined layout.....	62
Figure 4.9.	Global mesh of the computational domain for the four cylinder of inlined layout.....	62

LIST OF FIGURES**PAGE**

Figure 4.10.	Global mesh of the computational domain for the five cylinder of inlined layout.....	63
Figure 4.11.	Global mesh of the computational domain for the four cylinder of staggered layout.....	63
Figure 4.12.	Global mesh of the computational domain for the single cylinder of wavy fin configuration.....	64
Figure 4.13.	Global mesh of the computational domain for the four cylinder of staggered layout and wavy configuration.....	64
Figure 4.14.	Boundary conditions.....	65
Figure 4.15.	Time history of the velocity profile at the inflow section of the heat exchanger.....	68
Figure 4.16.	Schematic description of a typical PIV system (Akar, 2003)..	77
Figure 4.17.	A picture of the experimental set-up	79
Figure 5.1.	Grid independency check.....	83
Figure 5.2.	Comparison of experimental data of Seshimo and Fujii (1991) and the present numerical results of heat transfer coefficients as a function of Reynolds number.....	85
Figure 5.3.	Numerical results for the time averaged vorticity plots for the single cylinder configuration.....	88
Figure 5.4.	Local pressure distributions around the tube surface for the single row configuration with varying fin spacing and Reynolds number.....	92
Figure 5.5.	Average heat transfer coefficient as a function of fin spacing for Re=920.....	93
Figure 5.6.	Pressure drop coefficient between the inlet and outlet sections of the heat exchanger as a function of fin spacing for Re = 920.....	94
Figure 5.7.	$\overline{Nu}/(KF_s)$ as function of F_s for Re = 600 and Re = 2000 for the single cylinder configuration.....	96

LIST OF FIGURES

PAGE

Figure 5.8.	Average heat transfer coefficient on the fin surface as a function of number of tube rows for $F_s = 0.233$, $Re=1500$ and $Re=5000$, inlined arrangement.....	98
Figure 5.9.	Pressure drop between inlet and outlet sections of the heat exchanger for the multi row configuration as a function of number of tube rows at $F_s = 0.233$, $Re=1500$ and $Re=5000$, inlined layout.....	98
Figure 5.10.	The local pressure distributions around the tube surfaces for the multi row configuration at $Z=0.20$, for $F_s = 0.233$ at $Re_s = 600$, inlined and staggered layouts.....	100
Figure 5.11.	The variations of the local Nusselt number, Nu around the tube surface at $Z=0.20$, for $F_s = 0.233$ and $Re_s = 600$ for the 4 cylinder configuration; inlined and staggered layouts.....	102
Figure 5.12.	Average heat transfer coefficient as a function of Reynolds number for staggered and inlined layouts for the multi-row configuration(Misag-147, 003).....	103
Figure 5.13a.	Velocity vector, streamline, and heat transfer data at $Z=F_s/12$ plane around the first and the second cylinders for the wavy fin configuration at $Re=100$, laminar model results.....	107
Figure 5.13b.	Velocity vector, streamline, and heat transfer data at $Z=F_s/12$ plane around the first and the second cylinders for the wavy fin configuration at $Re=500$, laminar model results.....	108
Figure 5.13c.	Velocity vector, streamline, and heat transfer data at $Z=F_s/12$ plane around the first and the second cylinders for the wavy fin configuration at $Re=1000$, laminar model results.....	109
Figure 5.13d.	Velocity vector, streamline, and heat transfer data at $Z=F_s/12$ plane around the first and the second cylinders for the wavy fin configuration at $Re=1500$, laminar model results.....	110

LIST OF FIGURES**PAGE**

Figure 5.13e.	Velocity vector, streamline, and heat transfer data at $Z=F_s/12$ plane around the first and the second cylinders for the wavy fin configuration at $Re=1500$, RNG k-e model results.....	111
Figure 5.14a.	Velocity vector and streamline at $Z=F_s/12$ plane around the third and the fourth cylinders for the wavy fin configuration and staggered layout at $Re=100$, laminar model results.....	112
Figure 5.14b.	Velocity vector and streamline at $Z=F_s/12$ plane around the third and the fourth cylinders for the wavy fin configuration and staggered layout at $Re=500$, laminar model results.....	112
Figure 5.14c.	Velocity vector and streamline at $Z=F_s/12$ plane around the third and the fourth cylinders for the wavy fin configuration at $Re=1000$, laminar model results.....	113
Figure 5.14d.	Velocity vector and streamline at $Z=F_s/12$ plane around the third and the fourth cylinders for the wavy fin configuration at $Re=1500$, laminar model results.....	113
Figure 5.14e.	Velocity vector and streamline at $Z=F_s/12$ plane around the third and the fourth cylinders for the wavy fin configuration at $Re=1500$, RNG k-e model results.....	114
Figure 5.15a.	Velocity vector and streamline at $Z=F_s/2$ plane around the third and the fourth cylinders for the wavy fin configuration at $Re=100$, laminar model results.....	114
Figure 5.15b.	Velocity vector and streamline at $Z=F_s/2$ plane around the third and the fourth cylinders for the staggered layout, wavy fin configuration at $Re=500$, laminar model results.....	115
Figure 5.15c.	Velocity vector and streamline at $Z=F_s/2$ plane around the third and the fourth cylinders for the staggered layout, wavy fin configuration at $Re=1000$, laminar model results.....	115

LIST OF FIGURES

PAGE

Figure 5.15d.	Velocity vector and streamline at $Z=F_s/2$ plane around the third and the fourth cylinders for the staggered layout, wavy fin configuration at $Re=1500$, laminar model results.....	116
Figure 5.15e.	Velocity vector and streamline at $Z=F_s/2$ plane around the third and the fourth cylinders for the staggered layout, wavy fin configuration at $Re=1500$, RNG $k-\epsilon$ model results.....	116
Figure 5.16.	Pressure drop coefficient between the inlet and outlet sections of the heat exchanger as a function of Reynolds number obtained from Standard $k-\epsilon$ turbulence model simulation.....	118
Figure 5.17.	Pressure drop coefficient between the inlet and outlet sections of the heat exchanger as a function of Reynolds number obtained from RNG $k-\epsilon$ turbulence model simulation.....	118
Figure 5.18.	Heat transfer coefficient as a function of Reynolds number obtained from Standard $k-\epsilon$ turbulence model simulation.....	119
Figure 5.19.	Heat transfer coefficient as a function of Reynolds number obtained from RNG $k-\epsilon$ turbulence model simulation.....	119
Figure 5.20.	Friction factor as a function of Reynolds number obtained from Standard $k-\epsilon$ turbulence model simulation.....	120
Figure 5.21.	Friction factor as a function of Reynolds number obtained from RNG $k-\epsilon$ turbulence model simulation.....	120
Figure 5.22.	The time averaged vorticity contours around the tube surface at $F_s = 0.233$, in the $Z = 0$ plane, turbulence model results at $Re=4000$	122
Figure 5.23.	Time averaged local pressure distribution around the single cylinder, plain fin pattern obtained from turbulence model simulations.....	124

LIST OF FIGURES

PAGE

Figure 5.24.	Average Nusselt number and pressure drop coefficients across the heat exchanger as a function of Reynolds number for $F_s = 0.233$	126
Figure 5.25.	Colburn factor and Friction factor as a function of Reynolds number obtained for different turbulence model simulations for wavy, four cylinder staggered heat exchanger geometry for $F_s = 0.233$ (Misag 147, 2003).....	128
Figure 5.26.	Colburn factor and Friction factor as a function of Reynolds number obtained for Standard LES and Dynamic LES turbulence model simulations for wavy, four cylinder staggered layout heat exchanger geometry $F_s = 0.233$ (Misag 147, 2003).....	130
Figure 5.27.	Streamlines on the fin surface around the second and the third cylinder obtained from Standard LES simulations for the staggered wavy fin configuration at non dimensional time steps of 4.5, 9, and 22.....	132
Figure 5.28.	Streamlines on the fin surface around the second and the third row obtained from Dynamic LES simulations for the staggered layout wavy fin configuration at non- dimensional time steps of 4.5, 9, and 22.....	133
Figure 5.29.	Heat transfer coefficients on the fin surface around the second and the third cylinder obtained from Standard LES simulations for the wavy fin configuration at non-dimensional time steps of 4.5, 9, and 22.....	135
Figure 5.30.	Heat transfer coefficients on the fin surface around the second and the third cylinder obtained from Dynamic LES simulations for the wavy fin configuration at non-dimensional time steps of 4.5, 9, and 22.....	136

LIST OF FIGURES

PAGE

Figure 5.31.	Contours of vorticity magnitude at the centerline axis of the flow passage, at $Z=Fs/2$ plane, for the 4 cylinder, staggered layout, plain fin configuration obtained for Dynamic LES simulations, at non- dimensional time steps of 20, 40, and 80.....	138
Figure 5.32.	Contours of vorticity magnitude at the centerline axis of the flow passage, at $Z=Fs/2$ plane, for the 4 cylinder, staggered layout, wavy fin configuration obtained for Dynamic LES simulations, at non- dimensional time steps of 20, 40, and 80.....	139
Figure 5.33.	Instantaneous vorticity contours for transient simulations of the single cylinder configuration at different non-dimensional times of 2, 6, 17 and 52.....	142
Figure 5.34.	Location of theta-cut planes created around the tube surface..	144
Figure 5.35a.	Horseshoe vortex evolution in the circumference of the tube for the single cylinder case at various theta-cut planes at $\tau=2$.	146
Figure 5.35b.	Horseshoe vortex evolution in the circumference of the tube for the single cylinder case at various theta-cut planes at $\tau=6$.	147
Figure 5.35c.	Horseshoe vortex evolution in the circumference of the tube for the single cylinder case at various theta-cut planes at $\tau=17$	148
Figure 5.36.	Streamlines on the fin surface for the single cylinder case at $\tau=2, 6$, and 17	150
Figure 5.37.	Local heat transfer coefficients on the fin surface for the single cylinder case at $\tau=2, 6$, and 17	153
Figure 5.38.	Contours of vorticity magnitude for the staggered arrangement of the 4 cylinder configuration at $Z = 0.003$ plane at different non-dimensional times 8, 100, 400.....	155

LIST OF FIGURES

PAGE

Figure 5.39.	Heat transfer coefficients on the fin surface for the staggered arrangement of the 4 cylinder configuration at $Z = 0.003$ plane at different non-dimensional times at $\tau=8, 100, 400.....$	157
Figure 5.40.	Horseshoe vortex evolution in the circumference of the second cylinder for the staggered arrangement at various theta-cut planes at $\tau=100$ and $400.....$	159
Figure 5.41.	Comparison of experimental and numerical result for $F_s = 0.365$ at $Re=1460$. a) Time averaged velocity vector field (experimental result, side view) b) Time averaged vorticity contours (experimental result, side view) c) Time averaged streamline pattern (experimental result, side view) d) Time averaged streamline pattern (numerical study)	162
Figure 5.42	Experimental result for $F_s = 0.365$ at $Re = 600$ at $N = 6$, side view (a) instantaneous velocity vector field (b) instantaneous streamline pattern (c) instantaneous vorticity contours.....	165
Figure 5.43	Experimental result for $F_s = 0.365$ at $Re = 600$ at $N = 8$, side view (a) instantaneous velocity vector field (b) instantaneous streamline pattern (c) instantaneous vorticity contours.....	166
Figure 5.44	Experimental result for $F_s = 0.365$ at $Re = 1460$ at $N = 6$, side view (a) instantaneous velocity vector field (b) instantaneous streamline pattern (c) instantaneous vorticity contours.....	167
Figure 5.45	Experimental result for $F_s = 0.365$ at $Re = 1460$ at $N = 8$, side view (a) instantaneous velocity vector field (b) instantaneous streamline pattern (c) instantaneous vorticity contours.....	168
Figure 5.46	Experimental result for $F_s = 0.365$ at $Re = 4000$ at $N = 6$, side view (a) instantaneous velocity vector field (b) instantaneous streamline pattern (c) instantaneous vorticity contours.....	169

LIST OF FIGURES

PAGE

Figure 5.47	Experimental result for $F_s = 0.365$ at $Re = 4000$ at $N = 8$, side view (a) instantaneous velocity vector field (b) instantaneous streamline pattern (c) instantaneous vorticity contours.....	170
Figure 5.48	Experimental results obtained at the centerline of the flow passage for $F_s = 0.365$ at $Re = 600$, plan view (a) Time averaged velocity vector field (b) Time averaged streamline pattern (c) Time averaged vorticity contours.....	171
Figure 5.49	Experimental results obtained at the centerline of the flow passage for $F_s = 0.365$ at $Re = 1460$, plan view (a) Time averaged velocity vector field (b) Time averaged streamline pattern (c) Time averaged vorticity contours.....	172
Figure 5.50	Experimental results obtained at the centerline of the flow passage for $F_s = 0.365$ at $Re = 4000$, plan view (a) Time averaged velocity vector field (b) Time averaged streamline pattern (c) Time averaged vorticity contours.....	173
Figure 5.51	Experimental results obtained at the centerline of the flow passage for $F_s = 0.365$ at $Re = 600$, at $N=6$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	174
Figure 5.52	Experimental results obtained at the centerline of the flow passage for $F_s = 0.365$ at $Re = 600$, at $N=8$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	175
Figure 5.53	Experimental results obtained at the centerline of the flow passage for $F_s = 0.365$ at $Re = 1460$, at $N=6$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	176

LIST OF FIGURES

PAGE

Figure 5.54	Experimental results obtained at the centerline of the flow passage for $F_s = 0.365$ at $Re = 1460$, at $N=8$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	177
Figure 5.55	Experimental results obtained at the centerline of the flow passage for $F_s = 0.365$ at $Re = 4000$, at $N=6$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	178
Figure 5.56	Experimental results obtained at the centerline of the flow passage for $F_s = 0.365$ at $Re = 4000$, at $N=8$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	179
Figure 5.57	Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 600$, plan view (a) Time averaged velocity vector field (b) Time averaged streamline pattern (c) Time averaged vorticity contours.....	180
Figure 5.58	Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 1460$, plan view (a) Time averaged velocity vector field (b) Time averaged streamline pattern (c) Time averaged vorticity contours.....	181
Figure 5.59	Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 4000$, plan view (a) Time averaged velocity vector field (b) Time averaged streamline pattern (c) Time averaged vorticity contours.....	182
Figure 5.60	Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 600$, at $N = 6$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	183

LIST OF FIGURES

PAGE

Figure 5.61	Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 600$, at $N = 8$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	184
Figure 5.62	Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 1460$, at $N = 6$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	185
Figure 5.63	Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 1460$, at $N = 8$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	186
Figure 5.64	Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 4000$, at $N = 6$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	187
Figure 5.65	Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 4000$, at $N = 8$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	188
Figure 5.66	Experimental results obtained at the centerline of the flow passage for $F_s = 0.116$ at $Re = 1240$, plan view (a) Time averaged velocity vector field (b) Time averaged streamline pattern (c) Time averaged vorticity contours.....	189
Figure 5.67	Experimental results obtained at the centerline of the flow passage for $F_s = 0.116$ at $Re = 2500$, plan view (a) Time averaged velocity vector field (b) Time averaged streamline pattern (c) Time averaged vorticity contours.....	190

LIST OF FIGURES

PAGE

Figure 5.67	Experimental results obtained at the centerline of the flow passage for $F_s = 0.116$ at $Re = 2500$, plan view (a) Time averaged velocity vector field (b) Time averaged streamline pattern (c) Time averaged vorticity contours.....	190
Figure 5.68	Experimental results obtained close to the fin surface for $F_s = 0.116$ at $Re = 1240$, at $N = 6$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	191
Figure 5.69	Experimental results obtained close to the fin surface for $F_s = 0.116$ at $Re = 1240$, at $N = 8$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	192
Figure 5.70	Experimental results obtained close to the fin surface for $F_s = 0.116$ at $Re = 2500$, at $N = 6$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	193
Figure 5.71	Experimental results obtained close to the fin surface for $F_s = 0.116$ at $Re = 2500$, at $N = 8$, plan view (a) Instantaneous velocity vector field (b) Instantaneous streamline pattern (c) Instantaneous vorticity contours.....	194

LIST OF TABLES

PAGE

Table 5.1.	Comparison of heat transfer coefficients obtained for four-row plain, wavy, in- lined and staggered arrangements.....	103
Table 5.2.	Minimum and maximum vorticity magnitude obtained for Dynamic LES simulation for the staggered, 4 cylinder, plain and wavy fin configurations.	137



NOMENCLATURE

A	velocity amplitude
A_f	fin surface area (m^2)
A	surface area (m^2)
C_p	non-dimensional pressure coefficient
C_μ	Smagorinsky constant
C_{so}	model constant for the dynamic LES method (=0.15)
C_A	model constant for the dynamic LES method (=0.10)
c_v	specific heat at constant volume
d	tube diameter (m)
D	distance to the closest wall in the computational domain
e	internal energy
f	friction factor
F_s	non-dimensional fin spacing ($= s/d$)
F_p	fin pitch ($= F_s + t$)
G	filter function
h	local heat transfer coefficient ($\text{W}/\text{m}^2\text{K}$)
$\bar{h}$	average heat transfer coefficient ($\text{W}/\text{m}^2\text{K}$)
$H(x)$	Heaviside function
I	turbulence intensity (%)
j	Colburn factor
k	turbulent kinetic energy (m^2/s^2)
K	non-dimensional pressure drop coefficient
k_t	thermal conductivity
l	fin length (m)
ℓ	characteristic length scale of the turbulent flow
L	flow length (m)
L_s	length scale
L_m	mixing length for sub-grid scales

L_f	non- dimensional fin length
N	frame number
Nu	local Nusselt number
$\overline{Nu}$	overall Nusselt number
p	pressure (Pa)
p_n	local pressure (Pa)
p_∞	free stream pressure (Pa)
P	non-dimensional pressure
P_d	waffle height
$\overline{P}$	average non-dimensional pressure
$\dot{q}$	heat flux
$-\overline{\rho u_i u_j}$	Reynolds (turbulent) stress
$-\overline{q_j}$	Reynolds (turbulent) heat flux
q_j	filtered heat flux
Pr	Prandtl number
Pr_t	turbulent Prandtl number
R	specific gas constant
Re	Reynolds number
Re_D	Reynolds number based on the tube diameter ($=UD/\nu$)
Re_s	Reynolds number based on fin spacing ($=Us/\nu$)
s	fin spacing (m)
S	strain rate magnitude
S_t	distance between tube centers
S_h	heat sink term, K/s
S_{ij}	strain rate tensor
t	flow time (sec)
T	temperature (K)
$\overline{T}$	non-dimensional temperature
$\overline{u_i}$	time averaged (mean) velocity components in the x , y , and z direction

u'_i	fluctuating velocity component in the x , y , and z direction
U_∞	free stream velocity (m/s)
u_i	velocity components (u, v, w for $i = 1, 2$ and 3 respectively)
x_i	independent space coordinates (x, y, z for $i = 1, 2$ and 3 respectively)
V	volume of a computational cell (m^3)
ω_c	frequency (rad/s)
W	non-dimensional fin height
X	non- dimensional x - distance ($X = x/D$)
X_f	projected fin pattern length
Z	non-dimensional z - coordinate ($=z/s$)

Greek Symbols

ρ	fluid density (kg/m^3)
α	thermal diffusivity ($= k / (\rho C_p)$)
ΔP	pressure drop
ν	kinematic viscosity (m^2/s^2)
ν_t	turbulent kinematic viscosity (m^2/s^2)
τ	non- dimensional time ($= Ut/s$)
δ_f	fin thickness (m)
ε	turbulent dissipation rate (m^2/s^3)
Δ	averaged grid size
β	frequency parameter
μ	viscosity, $\text{kg}/(\text{ms})$
μ_t	turbulent (eddy) viscosity
μ_{eff}	effective viscosity ($\mu_{eff} = \mu + \mu_t$)
τ_{ij}	Reynolds stress tensor or SGS stress term or filtered stress
g	characteristic velocity of the turbulent flow

λ	a second viscosity related to the dynamic viscosity ($\lambda = -\frac{2}{3}\mu$)
φ	a flow property
$\hat{\Phi}$	filtered variable
θ	corrugation (wavy) angle ($^{\circ}$)
κ	von Karman constant
ϕ	cell centered value of a flow variable (field variable)
ω	time average square of the vorticity fluctuations
η	ratio between the characteristic time scales of turbulence and mean flow field

subscripts

S	based on fin spacing
in	at the inlet
w	at the wall
$exit$	at the exit
av	average
$crit$	critical
∞	free stream value

superscripts

$*$	non-dimensional value
$'$	fluctuating component
$\wedge$	filtered variable
$-$	time averaged (mean component)

1. INTRODUCTION

Technologies such as refinery, petrochemical, motor vehicle, drying, process, food, electronics, power production, manufacturing, and chemical, that recover heat and/or cooling energy reduce the cost and consumption of energy because the recaptured energy is potentially useful for heating and/or cooling applications: Waste heat from the exhaust and the engine cooling water of a natural gas or diesel engine driven electric generator can be recovered and used for water and space heating. Heat, recovered from the exhaust of a gas turbine, can produce steam that drives a turbo-generator. In commercial buildings, Cogeneration can also reduce the energy cost. Cogeneration units have a successful track record in a wide range of commercial applications for buildings that have high hot water requirements, such as restaurants, colleges, laundries, hotels and hospitals. In these buildings, using recovered heat to heat or preheat water is very cost-effective. Space and domestic water heating can be also accomplished by recovering the waste heat from a heat recovery system. All of the above-mentioned energy recovery systems involve a heat exchanger that is used to transfer of thermal energy between two or more fluids flowing in the system. Heat exchangers are also widely used in cooling of electronic devices and HVAC&R (heating, ventilating, air conditioning and refrigeration) applications where high heat transfer performance is obtained in a very small heat transfer surface area.

A heat exchanger can be defined as a heat transfer device that is used to transfer of internal thermal energy between two or more fluids available at different temperatures. In most heat exchangers the fluids are separated by a heat transfer surface, and ideally they do not mix (Kuppan, 2000).

According to Sherwin and Horsley (1996), there are three basic types of heat exchangers:

Recuperators- in which the two fluid streams are separated by a wall. The wall will generally be of metal, although a range of materials including ceramics, plastics and even paper can be used. Heat transfer within a recuperator takes place by means of

convection in the fluid streams either side of the wall and conduction across the wall itself.

Regenerators- in which the heat transfer takes place by means of heat storage device. The hot fluid flows through the device, thereby raising its temperature; then the fluid streams are switched and the cold fluid flows through the device. An alternative form of regenerator is a 'rotary regenerator'. A circular storage matrix rotates and alternately the hot and cold streams flow through the matrix. Such devices have been used in small gas turbine engines to preheat the air entering the combustion chamber using the exhaust gas.

Fluid contact heat exchangers- in which the two fluid streams are in direct contact with each other. A typical example is a cooling tower, in which water falls down through the tower and is cooled by a counterflow of air up through the tower. The two fluids are in direct contact and heat transfer takes place mainly by evaporation of the water.

Another classification is made by Kuppan (2000): In general, industrial heat exchangers have been classified according to (1) construction, (2) transfer processes, (3) degrees of surface compactness, (4) flow arrangements, (5) pass arrangements, (6) phase of the process fluids, (7) heat transfer mechanisms. These classifications are shown in Fig. 1.

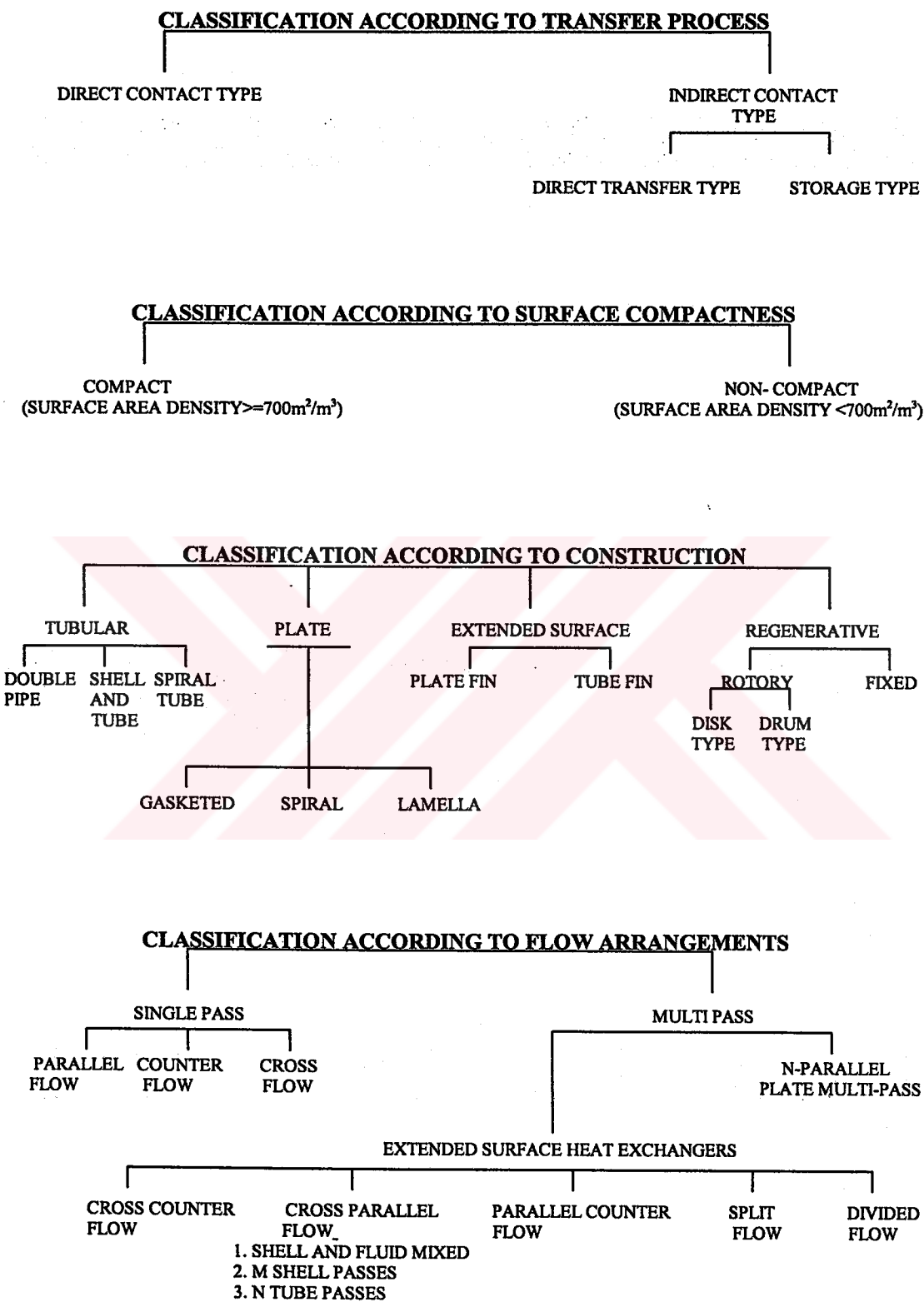


Figure 1.1. Classification of heat exchangers (Kuppan, 2000)

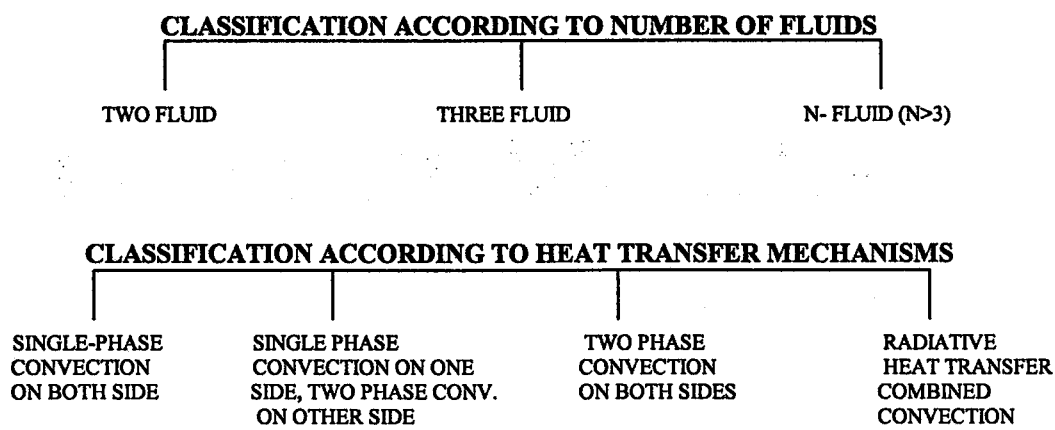


Figure 1.1. (continued) Classification of heat exchangers (Kuppan, 2000)

In the present investigation, the attention is focused on a compact, extended surface, cross-flow, single phase, two-fluid, plate fin heat exchanger with round tubes which has continuous fins attached to the outside of tube bundles. They employ a compact surface with low weight, support and structure; small volume; easy transport and higher thermal effectiveness compared to the other types. Therefore they are widely used in energy-intensive industries. They are also used in vehicular heat exchangers, condensers and evaporators in air conditioning and refrigeration industry, automotive radiators, aircraft oil coolers, unit air heaters, cooling of electronic devices, energy recovery, conservation and conversion, etc. These types of heat exchangers are generally referred to as *plate fin and tube*, *plate finned tube*, and *tube in plate fin* heat exchangers in the literature.

In the plate fin and tube heat exchangers, continuous and plain or specially configured fins are used on the outside of the array of tubes of staggered or inlined layouts passing perpendicularly through the plates. Fins are attached to the tubes by brazing, soldering, adhesive bonding, welding, mechanical expansion (press fit), or extrusion. Fins are primarily used to increase the surface area (when the heat transfer coefficient on that fluid side is relatively low) and consequently to increase the total rate of heat transfer. Generally water flows inside of the tubes and air is forced to flow through the fins. The dominant resistance is usually on the air side. The use of extended surfaces, or fins, reduce the air side thermal resistance. However, the resulting air side resistance may still exceed that of the liquid. In this case, it will be

advantageous to use specially configured extended surfaces which provide increased heat transfer coefficients compared to that for a plain fin. According to Kim et al. (1997), such special geometries may provide heat transfer coefficients 50-150 percent higher than those given by plain extended surfaces. The fins can be of a variety of geometries- plain, wavy, louvered or interrupted and the tubes can be circular, oval, elliptical, and rectangular as shown in Figure 1.2.

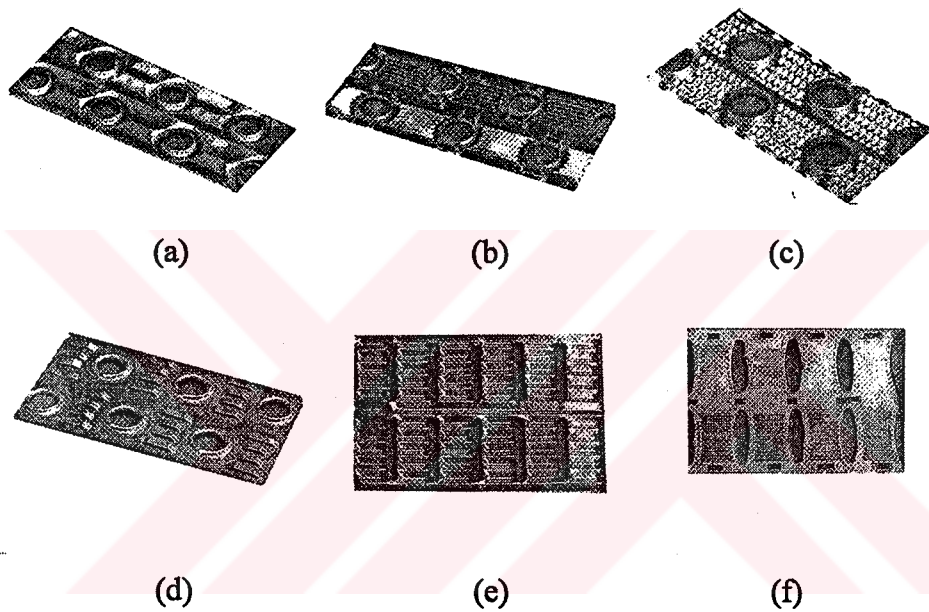


Figure 1.2. Flat or continuous fins on an array of tubes; (a) Wavy fin, (b) Multilouver fin, (c) Fin with structured surface roughness (circular dimples), (d) Parallel louver fin: all four fins with staggered round tubes, (e) wavy fin on inline flat tubes, (f) multilouver fins with inline elliptical tubes (Rohsenow et al, 1998).

In the industry, and in HVAC&R applications, mostly plain or wavy fin patterns and round tubes are used in the space heating and/or cooling devices such as in air conditioning units and fan-coils. Despite other fin configurations like wavy, slit, louver, convex-louver, etc., plain fin patterns are preferred because of their long term operation and lower friction characteristics. As indicated by Wang et al. (2000a), plain fin configuration is still the most popular fin pattern, owing to its simplicity, durability and versatility in application. However, wavy fins (or corrugated fins) are among the very popular fin patterns that were developed to

improve the heat transfer performance. The wavy surface can lengthen the path of the air flow inside the heat exchanger and cause better air flow mixing; consequently higher heat transfer performance is expected compared with the plain fin surface (Wang et al., 1997). Therefore, in the present investigation, it is thought to use both plain and wavy fin configurations and compare their heat transfer and fluid flow properties.

In the first part of the present study, various geometrical parameters are tested in order to determine an optimum characteristic dimensions for the plate fin and tube heat exchangers having plain fin configuration operating under laminar flow condition. In the second part, with the consideration of the turbulence effects, the best model predicting the air flow and corresponding heat transfer data is chosen. In the last part of the study, in order to consider time and space variations of turbulence effects, a new method is proposed and adapted to the numerical solution of the flow problem. The present numerical model is tested against experimental results available in the literature as well as it is validated by using PIV technique.

In Fig. 1.3, typical plate fin and tube heat exchanger configurations with plain and wavy fin patterns that are used in the present investigation can be seen. Characteristic geometrical parameters for a typical plate fin and tube heat exchanger may be classified as follows (Fig. 1.3):

d = tube diameter

s = fin spacing

δ_f = fin thickness

θ = corrugation angle (wavy angle)

F_p = fin pitch

P_d = waffle height

X_f = projected fin pattern length

S_t = distance between tube centers

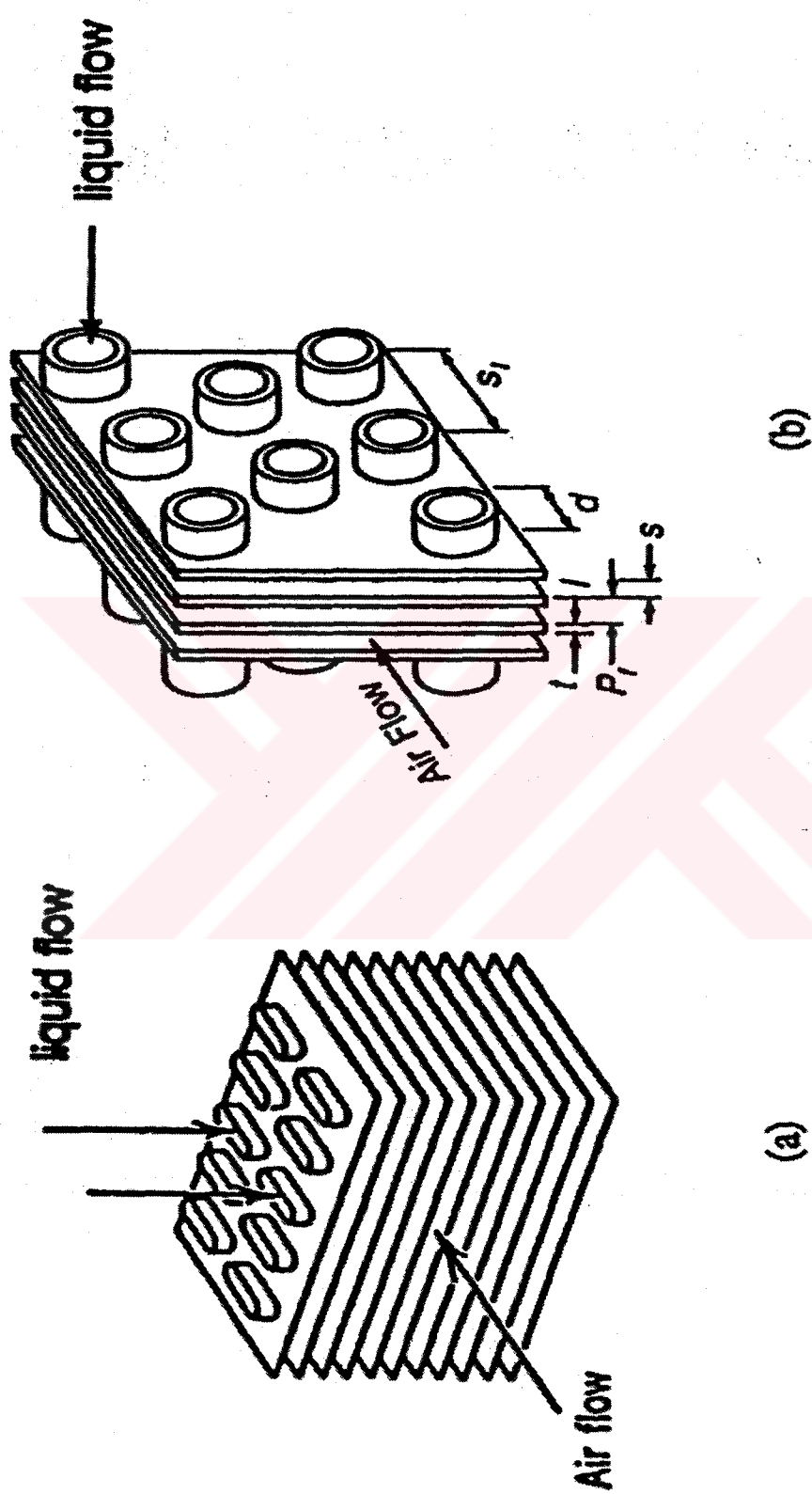
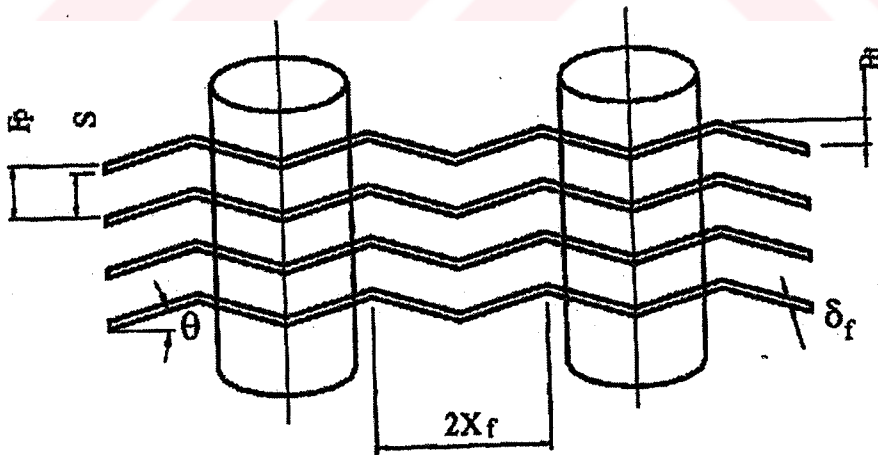
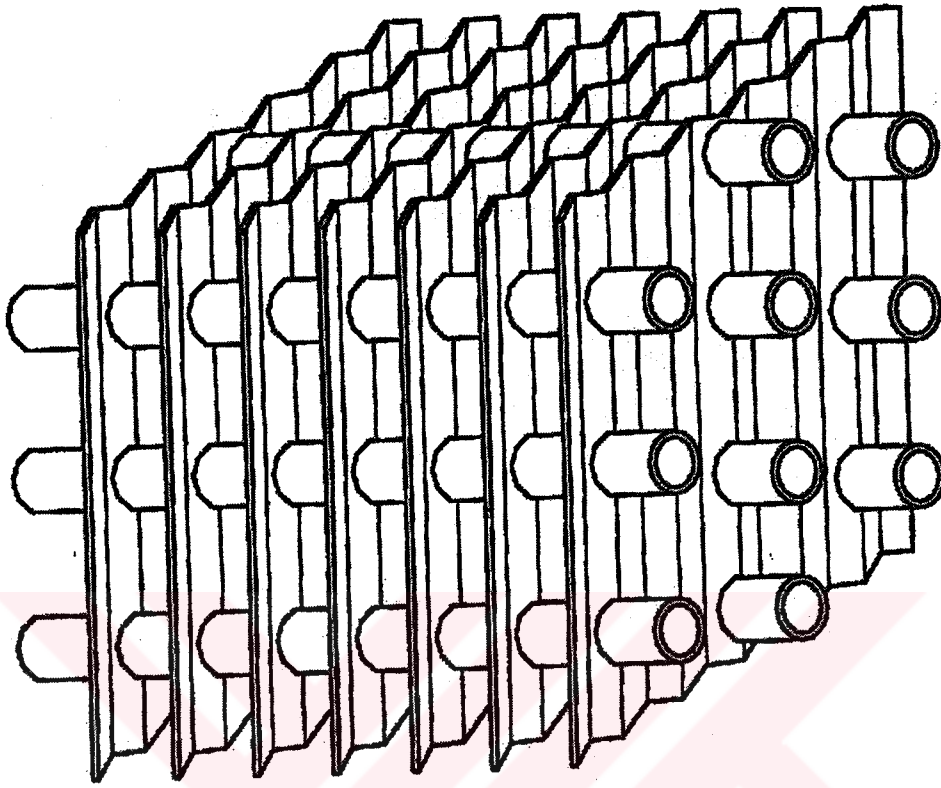


Figure 1.3. (a) Plain fin configuration with flat tubes (b) Plain fin configuration with round tubes (Kakac and Liu, 1997)



(c)

Figure 1.3. (continued) (c) Wavy (corrugated) fin configuration with round tubes (Wang et al., 1999)

2. BACKGROUND AND LITERATURE SURVEY

As it is well-known, the design of a heat exchanger involves a consideration of both the heat transfer rates between the fluids and the mechanical pumping power expended to overcome fluid friction (pressure drop) and move the fluids through the heat exchanger. The main design objective is to obtain the highest heat transfer rate per unit of the surface area with the least friction power expenditure. The friction power expenditure corresponds to mechanical energy in overcoming friction power generated in the heat exchanger as a result of fluid flow. In a plate fin and tube heat exchanger, the rate of heat transfer can be increased with increasing the flow velocity. Unfortunately, this is not a reasonable and a comfortable way since increasing the velocity will also increase the friction power expenditure, noise and vibration of the components of the heat exchanger where low-level noise is expected especially in occupied spaces. Instead of increasing the flow velocity, increasing the surface area of the heat exchanger may be another solution also results in an increase in the friction power expenditure. All of these considerations have led to the development of heat exchangers with large surface area density. An effective way to increase the surface area density is to introduce secondary surfaces, or fins. This involves compact heat transfer surfaces, or in other words high-performance surfaces, in which high heat flux is obtained relative to the friction power expenditure.

The compact heat exchanger considered in the present investigation employs a compact surface on the air side by its parallel fin plates passing perpendicularly through circular tubes. The air is forced to flow through the channels between these adjacent fins results in a complex three-dimensional flow. There are a number of experimental and numerical studies in the literature regarding flow and heat transfer properties of these types of heat exchangers. In most of the experimental studies turbulence effects did not considered and the experiments are conducted under laminar flow conditions. Similarly, most numerical studies are performed by laminar flow assumptions. As it is also stated by Shah et al. (1997), in the literature, there is no clear evidence yet that the flows in this type of heat exchanger passages are

unsteady laminar or low Reynolds number turbulent flows. According to Kundu et al. (1992), for plate fin and tube heat exchangers, the flow attains turbulent characteristics when the Reynolds number is larger than 400. Therefore in the present study it is decided to employ both laminar and turbulence models to investigate the flow structure and corresponding heat transfer data in the heat exchanger passage.

2.1. Geometrical and Flow Characteristics of Plate Fin and Tube Heat Exchangers

Flow characteristics of compact heat exchangers depend on the geometry of the exchanger. As shown in Fig. 1.3, the plate fin and tube heat exchangers consist of parallel plates passing perpendicularly through the tubes. The air is forced to flow through the channels between adjacent fins passing perpendicularly through the tubes.

Let's consider a bare tube bundle, without fins on the outside of the tubes, in which a fluid flows perpendicular to the tubes. In the absence of the fin plates, the flow would be two-dimensional over the tube bank. The bundle of tubes is a high-performance surface, since a new boundary layer builds on each tube resulting in much high heat transfer coefficients. Now let's consider the air flow through the channels formed by the parallel plate fins without the tubes. In the absence of the tubes, the air flow would be a simple two-dimensional flow with a developing velocity and thermal boundary layer at the channel entrance. With both tubes and fins present, the flow is highly three-dimensional with interacting secondary flows of opposite vorticity at the front and side of the tubes and a recirculating tube wake, all of which affect the fluid flow path and heating (FIDAP Users Manual, 2003).

The fins on the outside of the tubes are used in order to obtain high-performance surfaces. Kundu et al. (1992) stated that the use of specially configured fin geometries introduces further complications for the air flow in the heat exchanger passage. Madi et al. (1998) stated that the wavy fins accelerate the air flow and intensifies the turbulence of the air flow and enhance the heat transfer rate. Wavy fins form a corrugated flow passage in which boundary layer separation is induced

and a higher heat transfer coefficient at a given flow velocity is obtained. Boundary layers continuously created and destroyed due to the corrugation geometry resulting in improved air mixing (Tzanetakis et al., 2004). The surface waviness- induced periodic disruption and thinning of boundary layers along with lateral swirl mixing produce high local heat fluxes near the wall peak regions (Zhang et al., 2004).

Shah et al. (1997) reported the complexities of the flow structure in the fin and tube heat exchangers since it consists of swirl or flow separation, recirculation and reattachment. This complex three dimensional nature of the flow cause a pressure drop in the heat exchanger passage. According to Kim et al. (1999), the primary cause for pressure drop in the heat exchanger passage is the presence of cylinder array; the fins have less influence.

2.1.1. Definition of Horseshoe Vortex

When a flow passes through a wall-mounted object of sufficient size, it accelerates around the object and vertical velocity gradient of the flow is transformed into a pressure gradient on the leading edge of the structure. This pressure gradient results in a downward flow that impacts the object's base (Fig. 2.1). This downflow forms a vortex sweeps around and downstream of the object (Fig. 2.2). This vortex is termed as "horseshoe vortex" or "necklace vortex" due to its horseshoe shape. The horseshoe vortex can be seen in every day life at any time a boundary layer encounters a wall mounted bluff body obstruction and is very important in various engineering applications. It can be seen at an intersection of turbine blade and rotor; at a juncture of bridge pier and river bed; at a ground- building junction; at a wing-body junction of an airfoil; etc. It is a very dangerous in most of the engineering applications: Let's consider a bridge pier in a river bed (Fig.2.3). A bridge can fail due to the local scour at the base of the pier. Scour means enlargement of the cross section by the removal of boundary material due to the flow accelerations and vortices. Richardson (1995) described the development of the scour hole such that "The transport rate of sediment away from the vicinity of the base is greater than the transport rate into the region and consequently a scour hole develops". The

mechanism causing the scour formation is nothing but the horseshoe vortex system. The action of the horseshoe vortices removes bed material from around the pier nose. Knowledge of maximum possible scour around a bridge pier is of very important in safe and economic design of foundations of bridge piers. Figure 2.4 illustrates the area of horseshoe vortex at a cylindrical pier.

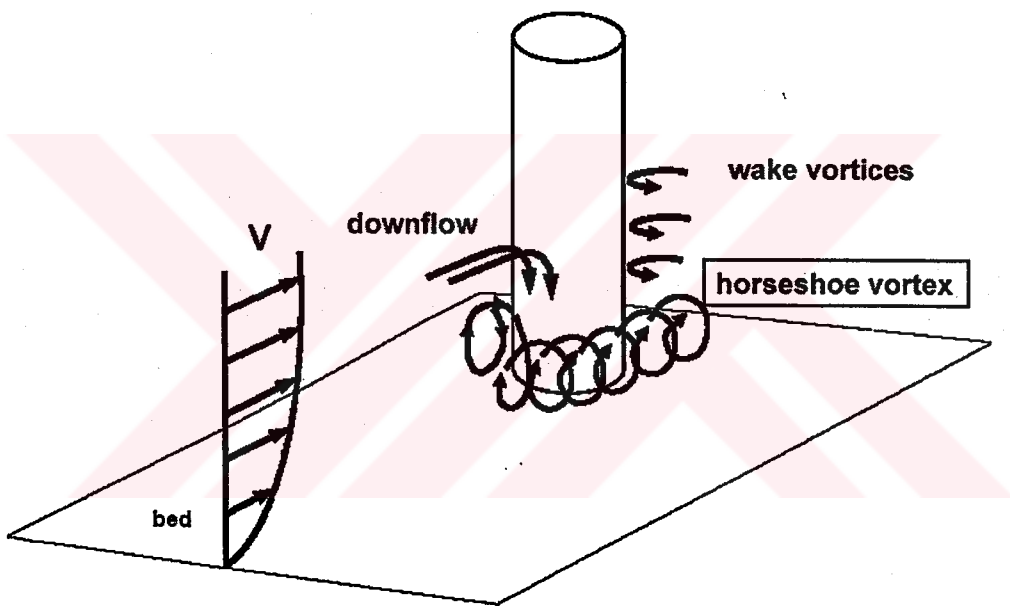


Figure 2.1. Local scour mechanism at the base of the bridge pier (Sheppard, 2004).

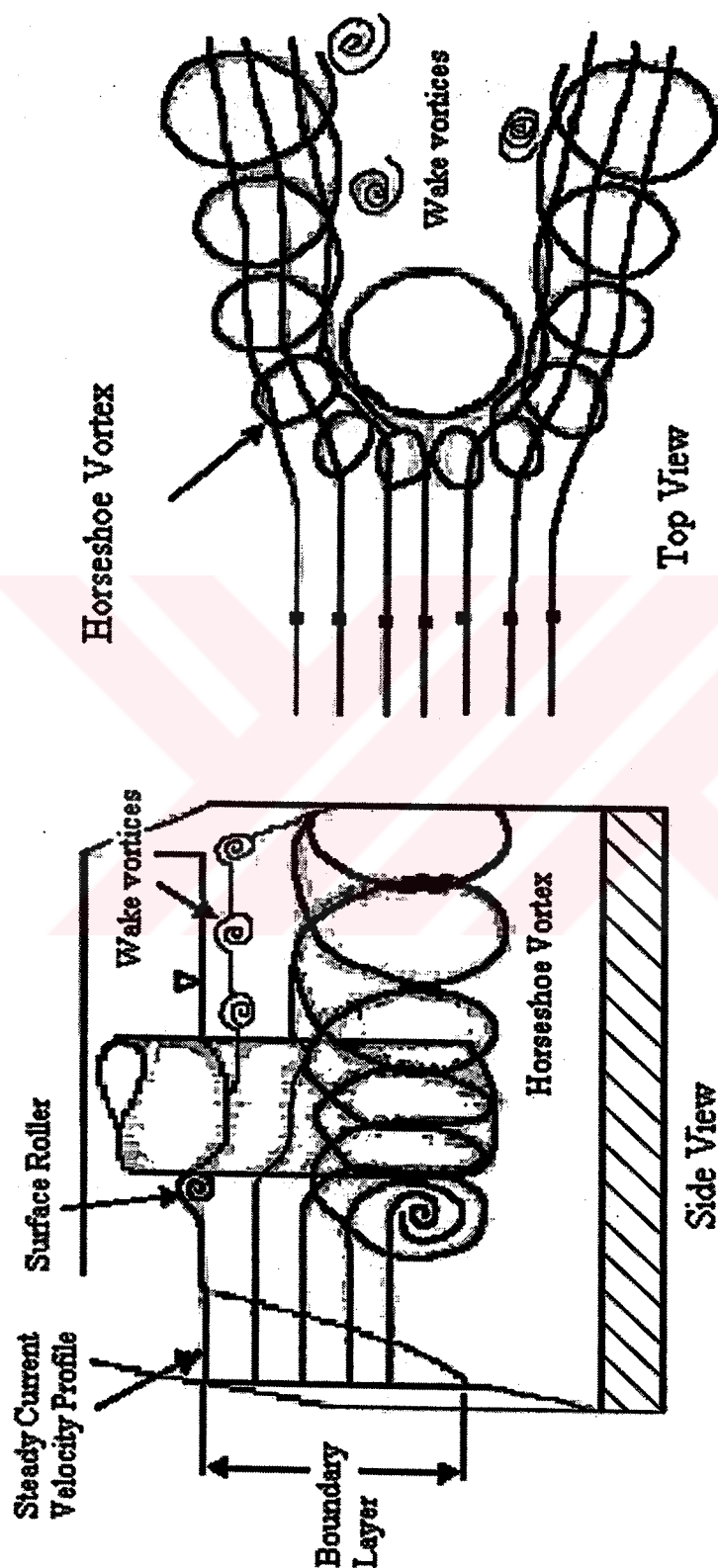


Figure 2.2. Flow patterns associated with local scour. At the water's surface, the flow and structure interaction forms a bow wave known as the surface roller. Beyond the points of flow separation on the structure, wake vortices occur (Miller, 2003).

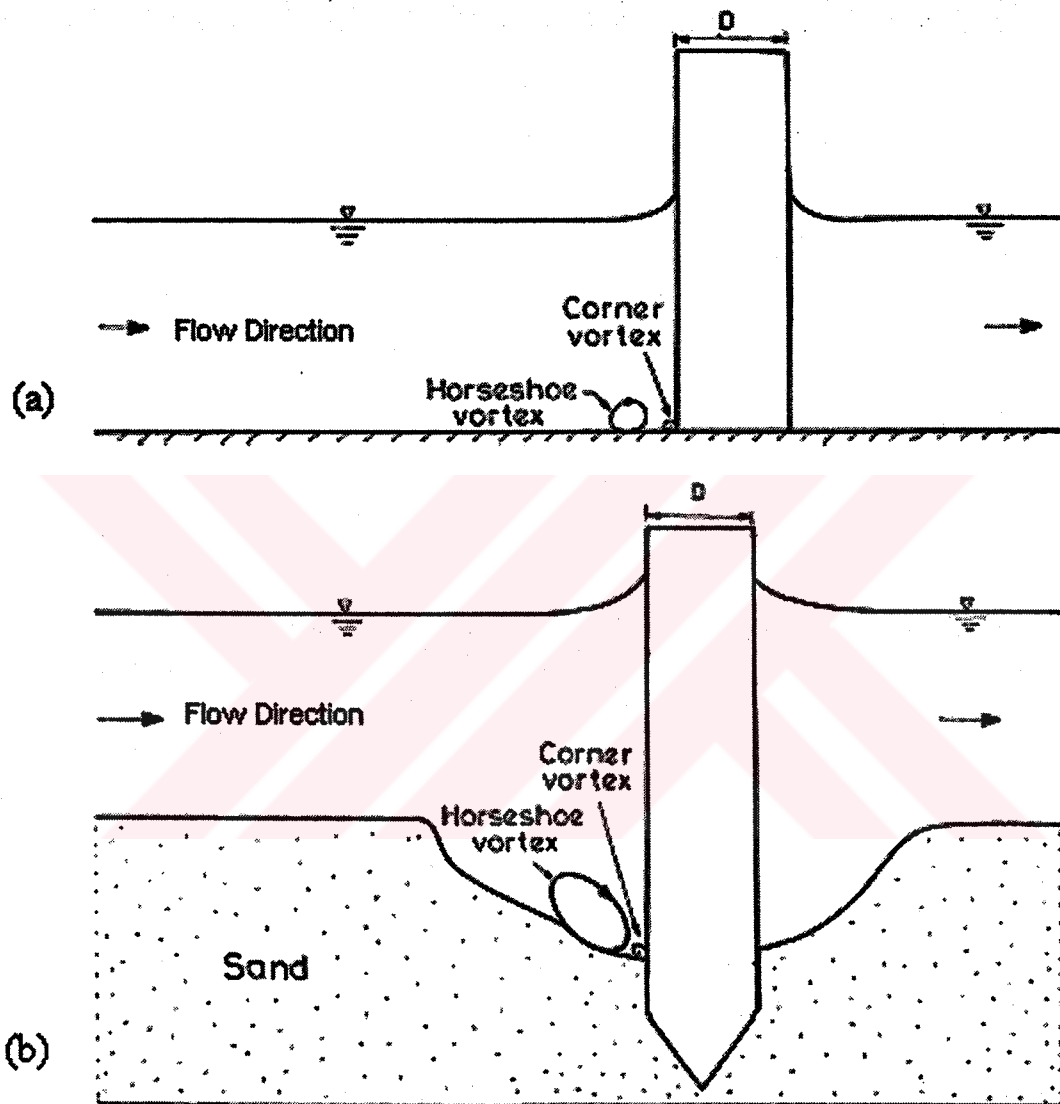


Figure 2.3. Sketches of a bridge pier in a river bed. (a) horseshoe vortex on rigid bed, (b) horseshoe vortex inside scour hole (Muzzammil and Ganngadhariah, 2003).

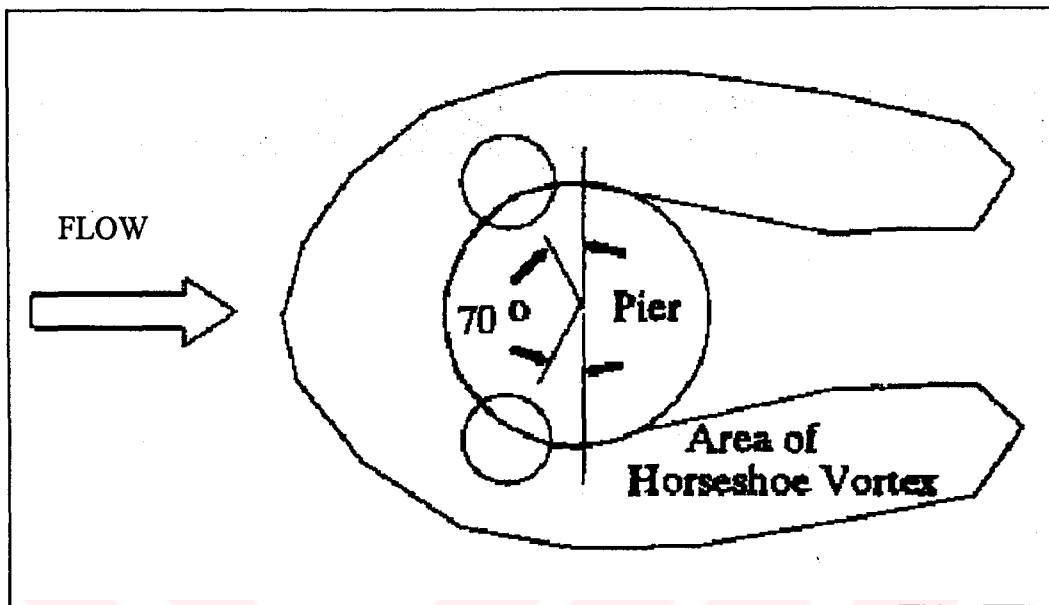


Figure 2.4. Schematic representation of the area of the horseshoe vortex (Fotherby, 1995)

The presence of horseshoe vortex system increases the local shear stresses, heat transfer, drag and noise, and modifies the lift characteristics of the profiles (Ballio et al., 1998). Although the horseshoe vortex system is very dangerous in most of the engineering applications, however, it is a desirable phenomenon in some applications such as in heat exchangers. In the heat exchangers, as the flow approaches the tubes, the horseshoe vortex system forms in front of each tube in the stagnation area near the tube and the fin junction region of the plate fin and tube heat exchangers and they introduce additional mixing of hot and cold fluids and consequently increase the fin heat transfer in that area. Kim et al. (1999) obtained increased heat transfer coefficients in front of the tubes and stated that the horseshoe vortex produced a natural heat transfer augmentation in the plate fin and tube heat exchangers. High heat transfer in front of the tube is also indicated by Valencia et al. (1996). Tsai and Sheu (1998) indicated a vortical core line occurs around the tube when the core of each horseshoe vortex is connected to each other. Kim and Song (2002 and 2003) also examined horseshoe vortex mechanism for a one and a two

cylinder plate fin and tube heat exchanger and found high heat and mass transfer coefficients in front of the tubes due to the existence of a horseshoe vortex.

2.1.2. Boundary Layer Separation and Flow Around a Circular Cylinder

In the plate fin and tube heat exchangers, a very complex flow characteristics can be seen due to the presence of a number of plates and tubes. A new boundary layer is created on each cylindrical tube in the tube bundle. In addition, the boundary layer is continuously created and destroyed by the presence of corrugated plate fins. Therefore the boundary layer and its separation as well as flow around a single cylinder concepts should be well understood in the context of the optimum design of plate fin and tube heat exchangers.

In case of an incompressible and a viscous fluid flow, the presence of the fluid viscosity slows down the fluid particles very close to the solid surface and forms a thin slow-moving fluid layer called a boundary layer. The flow velocity is zero at the surface to satisfy the no-slip boundary condition. Inside the boundary layer, flow momentum is quite low since it experiences a strong viscous flow resistance. After the fluid enters a duct, the boundary layer gradually fills the entire duct cross section and the flow then is said to be fully developed.

In case of flow around a stationary circular cylinder, the impingement of the flow on the cylinder creates a stagnation point on the approaching surface in front of the cylinder. Because of the momentum balance, a no-slip condition exists everywhere on the surface of the cylinder, i.e., the velocity must vanish everywhere on the surface. Consequently a boundary layer is created where the velocity transitions from a value of zero at the surface to the free stream value some distance away from the cylinder surface. The thickness of the boundary layer increases towards the downstream direction. The flow in the boundary layer is first laminar and becomes unstable from a certain point. The thickness of the boundary layer increases after transition to turbulent flow.

When a fluid particle moves within the boundary layer adjacent to the surface of the cylinder, its motion depends on the several forces acting on it: It is subjected to

frictional force at the surface, viscous force by the neighboring flow stream, and pressure force depending on the external flow condition. The fluid can continue its forward motion if the pressure gradient is favorable. However, if the fluid experiences an adverse pressure gradient, that is when the pressure force is against the motion of the fluid, the fluid particles can be stopped and sometimes be forced to move backward. The retardation and backward-moving motion always occur at the surface since the friction force exerted by the wall is greatest and the fluid momentum is much lower there. The inertia of the fluid as it rounds the top and the bottom of the trailing surface causes the flow to separate at these locations.

The flow separation prevents the pressure being recovered at the rear half of the object, therefore, causing a difference in pressure between the front and the back sides of the object. The pressure is maximum at the stagnation point and gradually decreases along the front half of the cylinder. The flow stays attached in this favorable pressure region as expected. However, the pressure starts to increase in the rear half of the cylinder and the particle now experiences an adverse pressure gradient. Consequently, the flow separates from the surface and creating a highly turbulent (unsteady) region behind the cylinder called the wake. The pressure inside the wake region remains low as the flow separates. The pressure difference in front and at the rear of the tube causes a net drag force (pressure force) acting on the body as is usually called the “form drag” or “pressure drag”.

The shear stress in the boundary layer region is very high due to the high velocity gradients. Within the boundary layer, governing equations of motion are Navier- Stokes, continuity and energy equations. After the flow separation from the surface of the cylinder occurs, the boundary layer equations become invalid and the full solution of the Navier-Stokes Equations is required in order to resolve the flow in the separated region. The point of separation can be defined as the point where the normal gradient of the tangential velocity vanishes so that the wall shear stress is zero. The location of the separation point for a circular cylinder is not fixed and determination of it is a major topic of fluid mechanics especially in case of oscillating cylinders.

The unsteady nature of the wake flow is described extensively in Shih (1999). Valencia et al. (1996), Tsai and Sheu (1998) and Kim et al. (1999) indicated that, in the plate fin and tube heat exchangers, a very -low- performance region exists behind the tube, and low heat transfer coefficients in the tube wake after the flow separation occurs.

2.2. Fluid Flow Topology

Fluid flow topology can be described as visual representation of vector and tensor fields and can provide useful information on the structure of the flow field and identification of critical points such as nodes and saddle points, foci and line of separation.

In the velocity, shear stress, or vorticity vector fields, singular points are classified as nodes and saddle points. Nodes are further classified into nodal points and foci. A nodal point is a point common to an infinite number of streamlines. If the streamlines converge to the nodal point, it is said to be a nodal point of separation. If they diverge from the nodal point it is said to be a nodal point of attachment. If an infinite number of streamlines spiral around the node, then the node is said to be a focus. If the streamlines away from the node, they are defined as foci of attachment. If they are spiral into the node, they are defined as foci of separation. If there are only two lines normal to each other through the point, the point is called a saddle. Figure 2.5 shows the singular points defined in this section.

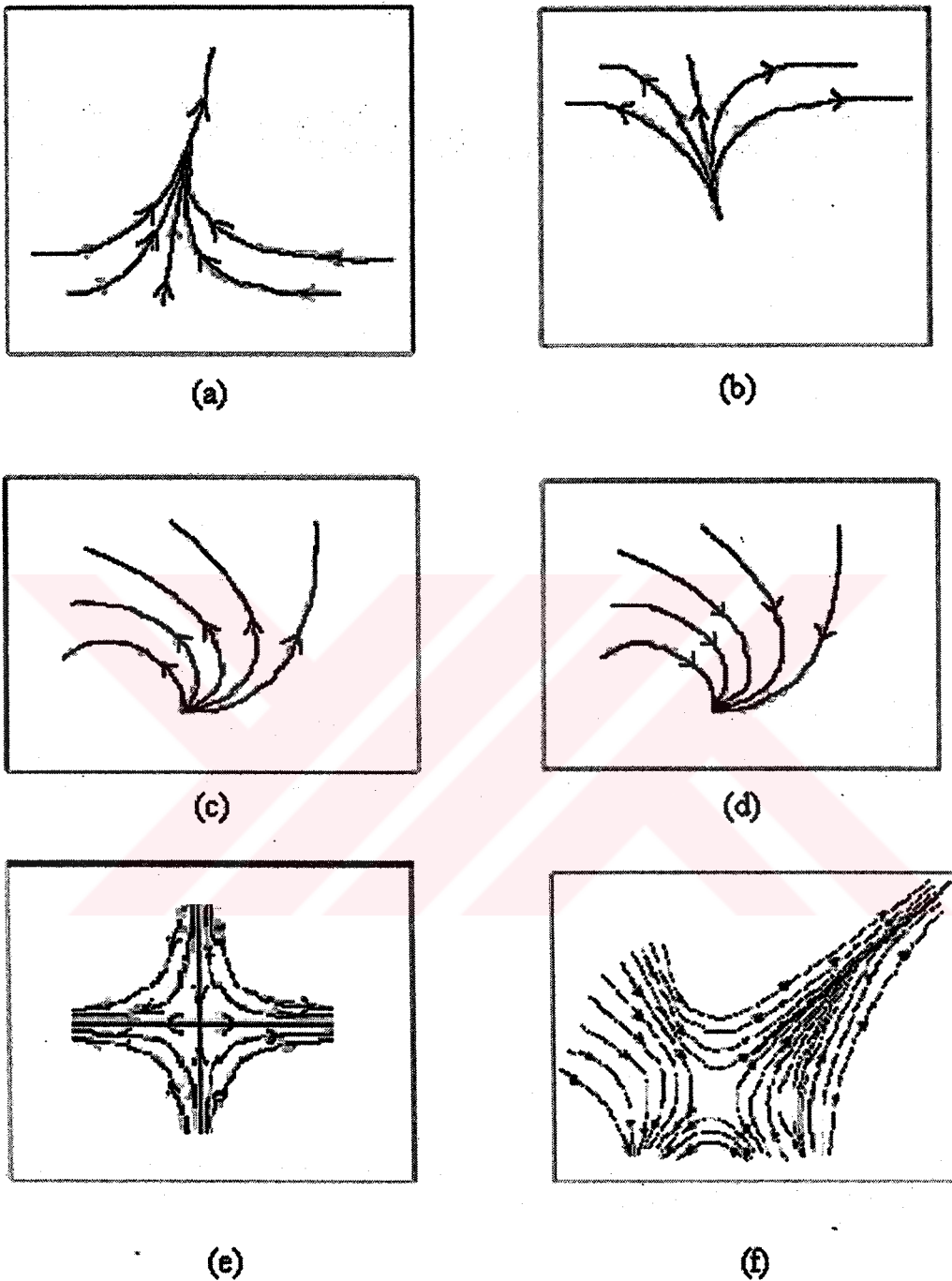


Figure 2.5. Singular points. a) Nodal point of separation, b) Nodal point of attachment, c) Foci of attachment, d) Foci of separation, e) Saddle point, f) Line of separation, (Waclawicz, 2001).

As reported by Waclawicz (2001), a line of separation is present when the streamlines emerging from the nodal points of attachment are prevented from crossing by the presence of a particular streamline emerging from the saddle point as seen in Fig. 2.5f.

In order to well-understand above mentioned singular points, a sketch shown in Fig. 2.6, flow around a surface mounted cubical obstacle, is shown. The horseshoe vortex formed in front of the cube and singular points in the flow domain around the cube can be clearly seen from the figure.

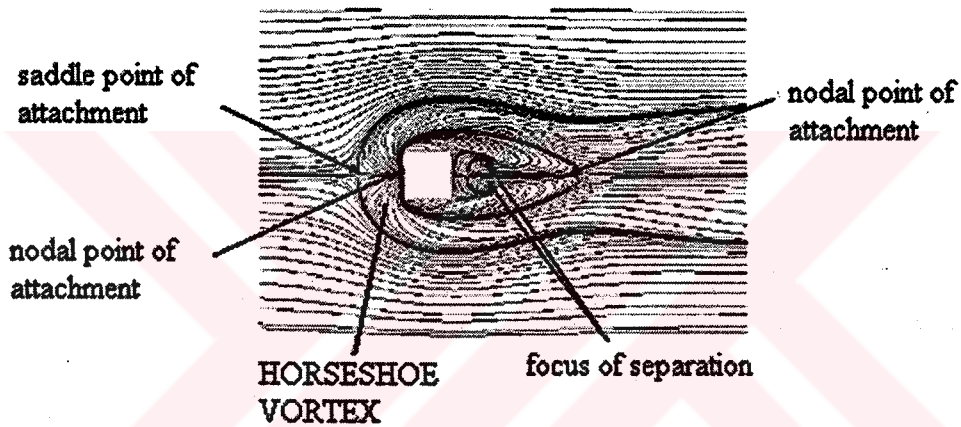


Figure 2.6. Flow around a surface mounted cubical obstacle (Xing and Baili, 2004)

In the present study, singular points are observed throughout the flow field. For better understanding of the local flow details and corresponding heat transfer characteristics of the plate fin and tube heat exchangers, singular points are sketched and they are explained in accordance with the heat transfer coefficients and vortex formation mechanisms in the Results and Discussion section.

2.3. Literature Survey for Plate Fin and Tube Heat Exchangers

In the literature there are a number of studies regarding flow and heat transfer characteristics of plate fin and tube heat exchangers with various fin and tube configurations. In this dissertation only plain and wavy fin and circular tube configurations are investigated in terms of heat transfer and fluid flow characteristics. Therefore heat exchangers having different configurations such as louvered fins, elliptical tubes, etc. will not be addressed here. The studies in the literature may be classified as experimental and numerical studies. Some authors combine their experimental and numerical data in a single study and compare their results. Therefore the literature survey is presented under two different categories considering experimental and numerical studies.

2.3.1. Experimental Studies

Saboya and Sparrow (1974) measured local heat transfer coefficients using naphthalene mass transfer technique and observed high values of heat and mass transfer coefficients in the forward part of the fin due to the presence of developing boundary layer and low heat and mass transfer coefficients in the region downstream of the tube. They observed that a vortex develops which sweeps around the tube and produces a natural heat transfer augmentation.

Goldstein and Sparrow (1976) measured local and average heat transfer coefficients for a single row wavy fin and tube heat exchanger with the naphthalene sublimation technique in the Reynolds number range of $135 < Re < 1200$ and compared their results with a plain fin configuration. They indicated that there is a 45% increase in the average heat transfer coefficient at $Re=1000$ for the wavy fin pattern compared to the plain one.

Rosman et al. (1984) measured overall heat transfer coefficient for single and two cylinder plate fin and tube heat exchanger and performed 2-D numerical heat transfer analysis in order to obtain temperature distribution and fin efficiency for the range of Reynolds number $200 < Re < 1700$. They found the fin efficiency was higher

for the two cylinder case due to the fact that isothermal situations are achieved and there was no influence of entrance region

Seshimo and Fujii (1991) investigated effect of ring depression, fin spacing, fin length, and tube row number on the heat transfer and friction performance of heat exchangers with plain fin configurations in the $300 < Re < 2000$ range. They determined a new characteristic velocity and length to correlate the experimental results and obtained a Nusselt number and friction factor relation for these type of heat exchangers.

Kundu et al. (1992) measured heat transfer and pressure drop data for laminar and turbulent flow over an inline cylinder array for eight cylinders with three different aspect ratios (ratio of fin spacing to cylinder diameter) in the $220 < Re < 2800$ range concerning flow transition from laminar to turbulent flow regime.

Jang et al. (1996) have performed experimental studies over a 3-D multi-row (1-6 cylinders) plate fin and tube heat exchanger to investigate effects of tube arrangement, tube row number and fin pitch (8-12 fins per inch) for fin spacing-based Reynolds number in the range of $60 < Re < 900$. The experiments were conducted using the wind tunnel. The average heat transfer coefficient of the staggered arrangement was 15-27% higher than that of the inlined arrangement, while the pressure drop of the staggered arrangement was 20-25 % higher than that of the inlined arrangement.

Wang et al. (1997) conducted several tests on wavy fin and tube heat exchangers with different geometrical parameters including the number of tube rows, fin pitch, and also tube arrangement for $400 < Re < 8000$ range in the wind tunnel. They observed that for the staggered layout at higher Reynolds numbers, downstream turbulence causes a higher heat transfer coefficient with an increase in the number of tube rows. Also the heat transfer coefficients for wavy fin geometry show a 55% to 70% increase compared to the plain fin geometry while the corresponding values for the friction factor was 66% to 140%.

Kim et al. (1997) used a multiple regression technique to correlate wavy fin geometries for both staggered and in-line arrays of circular tubes with varying number of tube rows, tube diameter, fin spacing and distance between tubes in the

direction either normal to the flow or parallel to the flow. In the literature, there are also a number of experimental studies regarding empirical correlations to describe the performance of the plate fin and tube heat exchangers in terms of heat transfer and flow friction characteristics. Some examples are: Wang et al. (2002), Wang et al. (2000a), Wang et al. (2000b), Wang, (1999b), Wang (1999c), Kim et al. (1999), Wang et. al. (2000c).

Rocha et al. (1997) determined heat transfer coefficients experimentally for circular and elliptical tube configurations for one and two cylinder tubes and they obtained 18% fin efficiency gain for elliptical tubes as compared to the circular one.

Madi et al. (1998) conducted wind tunnel tests to investigate effects of fin thickness and number of tube rows on the plain and wavy heat exchanger performance. They found increased heat transfer and friction characteristics for the wavy fin pattern compared to the plain one. They outlined the wavy fins accelerate air flow and intensifies the turbulence of the air flow between them, enhancing the heat transfer rate. They found that the effect of number of tube rows has negligible effect on the friction factor and that the fin thickness effect on the friction factor was found to be very small while the j factor increased with the decrease of fin thickness.

Wang et. al. (1998) investigated the effects of number of tube row and fin pitch on the thermal- hydraulic characteristics for wavy fin patterns for the Reynolds number based on the tube collar diameter in the $300 < Re < 8000$ range. Tests were conducted in the open wind tunnel. They concluded that the effect of number of tube rows on heat transfer performance changes significantly with the fin pitch. The amount of this effect was changing according to the Reynolds number. For $Re > 2000$, the effect of the tube row number diminishes as a result of vortex shedding caused by the tube row.

Critoph et al. (1999) measured local heat transfer coefficient on the fin and compared steady state and transient measurement results conducted by using liquid crystal thermography. Authors concluded that the transient method did not give results consistent with the steady state method when heat transfer coefficients are based on the fluid inlet temperature.

Wang et al. (1999a) studied the effects of the number of tube rows, fin pitch and edge corrugation on the performance of fin- and- tube heat exchangers having wavy fin configurations in an open wind tunnel test unit in the $500 < Re < 10^4$ range. Authors used 22 test samples with varying geometrical parameters.

Wang et al. (1999d) examined effect of waffle height on wavy plate fin and tube heat exchangers and compared plain and wavy fin patterns in terms of heat transfer and pressure drop characteristics. They founded larger heat transfer coefficient and pressure drop for the wavy fin pattern compared to those for plain fin pattern. It was shown that both heat transfer coefficient and pressure drop was increased as the waffle height was increased. The heat transfer performance for a larger waffle height was more strongly dependent on the fin pitch. The effect of fin pitch on the heat transfer performance is comparatively small for a small waffle height.

Yan and Sheen (2000) conducted wind tunnel experiments to determine effects of fin patterns having plain, wavy and louver fin surfaces on the performance of plate fin and tube heat exchangers in the $300 < Re < 2000$ range for varying tube row number and fin spacing. They found that the wavy fin pattern shows the highest performance for $Re < 1500$.

Mendez et al. (2000) examined effect of fin spacing on a single cylinder heat exchanger with plain fin configuration in the $260 < Re < 1460$ range through dye visualization technique. They showed that the horseshoe vortex development depends on the fin spacing and corresponds to the peak in the Nusselt number.

Halici et al. (2001) conducted experiments to determine the effects of number of tube rows on the performance of plate fin and tube heat exchangers for $300 < Re < 2000$. They found the Colburn and friction factors and heat transfer coefficients were decreased with the increase in the tube row number.

Lozza and Merlo (2001) carried out wind tunnel experiments to determine performance of various fin surface geometries including plain, wavy, louvered and winglet types.

Ay et al. (2002) performed experimental study using infrared thermovision to determine global temperature distribution over the fin surface. The local heat transfer

coefficients over the fin were determined by means of a control volume based finite difference formulation. The average heat transfer coefficient for the staggered configuration is 14-32% higher than that of in- lined configuration.

Wierzbowski and Stasiek (2002) used liquid thermochromic liquid crystal technique to determine local heat transfer data for a flat plate heat exchanger with in- line, staggered tubes and with vortex generators. They reported also the transitional range of Reynolds number to be in the $1000 < Re < 2500$ range for these types of heat exchangers.

Kim and Song (2002) studied the effects of distance between plates and tube placement for a single tube row in the $114 < Re < 2660$ range using naphthalene sublimation technique. They observed a subsidiary horseshoe vortex when the Reynolds number is as large as 2660. They found that total mass transfer rate is increased up to 25% by proper positioning of the tube.

Kim and Song (2003) observed that the horseshoe vortex formed in front of the first tube extends towards the second tube and increases the fin heat transfer rate in front of the both tubes. They suggested a non- dimensional optimal fin spacing to be equal to $s/D=0.5$ for the $1770 < Re < 2660$ range considering both heat transfer and pressure drop measurements.

2.3.2. Numerical Studies

Wung and Chen (1989) solved numerically the convective motion in staggered and in- lined tube arrays by the Finite Analytic Method. Solutions for Reynolds number of 40, 120, 400, and 800 were obtained. Differences in stream function, vorticity function, and location of separation and reattachment for flow past in- line tube arrays and staggered tube array were predicted and compared. The zone of separation for both arrays tends to increase with increasing Reynolds number. The predicted results on flow field and heat transfer were shown to agree with available experimental measurements. In the second part of their study, (Wung and chen, 1989), authors solved the same problem to determine the temperature field, heat transfer characteristics and pressure drop again by the Finite Analytic Method.

The heat transfer in a staggered array of tubes was found to be higher than that in an in-lined array of tubes. However, the pressure drop in a staggered array arrangement was also higher. Local heat transfer varied between both arrays and depended on Reynolds number and Prandtl number. At high Reynolds number, the local heat transfer tended to peak at the upstream surface of the heated tube but away from the stagnation point and became minimum at the separation point. Heat transfer in recirculation zones were, in general, small.

Kundu et al. (1991a) calculated heat transfer data for a 2-D heat exchanger module composed of inline cylinders between two parallel plates using an implicit finite difference method and reported influence of spacing between the cylinders on the heat transfer coefficient at low Reynolds numbers over a 4 inlined cylinders between two parallel plates.

Kundu et al. (1991b) solved governing flow and energy equations numerically for 2-D, incompressible, laminar flow at $50 < Re < 500$ range for a four cylinders and reported three different separation patterns depending on the spacing between cylinders and plates: A stable separated flow in which standing vortices are obtained when the fin spacing was small; a temporal and periodic separated zone in which the vortices are shed periodically at larger fin spacing; and a local separated flow that does not extent to the next tube in which length of the recirculation zone is less than the fin spacing.

Kundu et al. (1992) supported their experimental measurements for heat transfer and pressure drop data for laminar and turbulent flow with numerically computed data obtained for laminar flow. Their numerical and experimental data agreed well for low Reynolds numbers while they depart from each other at larger Reynolds number due to the fact that the numerical data for laminar flow do not include the turbulent mixing effect.

Valencia et al. (1996) carried out wind tunnel experiments to investigate effect of round and flat tubes on the heat transfer performance at $Re = 2440$ and recommended the location of the vortex generator for both tube shapes.

Jang et al. (1996) have performed numerical computations using the SIMPLER algorithm and have compared the performance of inlined and staggered

tube arrangements. The average heat transfer coefficient of the staggered arrangement was 15-27% higher than that of the inlined arrangement, while the pressure drop of the staggered arrangement was 20-25 % higher than that of the inlined arrangement. The numerical predictions agreed well with experimental data.

Jang et al. (1997) studied effects of tube row number, wavy angle, wavy height for Reynolds number ranging from 400 to 1200 by using SIMPLER algorithm. They concluded that row effect is less important in a wavy fin compared to the plain fin. They obtained the highest flow area goodness factor for wavy angle and wavy height is 8.95° and 1.5 mm, respectively.

Barsamian and Hassan (1997) performed simulations using the GUST computer program in which the large eddy simulation technique is adopted for time dependent 2-D tube bundle arrays of staggered and in-lined geometries without fins.

Shah et al. (1997) reviewed advances in numerical analysis for all types of compact heat exchangers in terms of fluid flow, heat transfer and flow friction characteristics. They pointed out for accurate prediction of local and overall Nusselt number for these heat exchangers will only be possible by analyzing time- dependent 3-D flows. They summarized an importance of unsteady simulations stating that: Unsteady flow simulation is very promising as it gives local instantaneous information; however, to be confident in the results of a numerical simulation, a careful validation should be provided on instantaneous data and not only on the time average values. Also most of the flows encountered are unsteady, often with flow separation and recirculation zones, and the Reynolds number range is typically $100 < Re < 2000$ for compact heat exchanger surfaces. The separation and reattachment of the flow and vortices in the wake of the fins affect both heat transfer and pressure drop, and these mechanisms cannot be predicted with time average modeling. Unsteady laminar model requires a very fine description of the geometry and the computation capacity limits its application to 2-D modeling. There is no clear evidence yet that the flows in this Reynolds number range are unsteady laminar or low Reynolds number turbulent flows; although it is possible that gradually the unsteady laminar flows associated with the interruptions become low Reynolds number turbulent flows. Hence, more sophisticated experimentation on small sample

fin geometries is required to determine the flow characteristics. At the same time, LES and DNS methods should be further developed to simulate the local flow and heat transfer characteristics of compact heat exchanger surfaces.

Rocha et al. (1997) found 18% fin efficiency gain for elliptical tubes as compared to the circular one and were compared their numerical results for fin efficiency for elliptic tubes with published data for circular tubes.

Tsai and Sheu (1998) conducted a 3-D numerical study by using a SIMPLER based solution algorithm in the $83 < Re < 258$ range to show regions of high and low heat transfer coefficients in a flow passage consists of 2 cylinders of staggered layout and a plain fin. The same authors published similar results for a wavy fin configuration in another publication which is Tsai and Sheu (1999).

Mendez et al. (2000) studied fin spacing effects in the $260 < Re < 1460$ range by using FIDAP finite element code. They derived new expressions for overall Nusselt number and pressure drop in the context of optimization of plate fin and tube heat exchangers. They found the optimum fin spacing to be in the $0.128 < S/D < 0.240$ range for $Re=630$

3. OBJECTIVES AND OUTLINE OF PRESENT INVESTIGATION

As it is outlined in the previous sections, the heat exchanger flow passage that consists of plain or specially configured parallel channels located on the outside of the tube bundle results in a very complex geometry. This complex geometry causes formation of a highly complex and three dimensional air flow and heat transfer characteristics. This complex flow behavior includes developing velocity and thermal boundary layers at the channel entrances; continuous creation, destruction, separation and reattachment of new boundary layers due to the presence of the wavy fins and due to the existence of large amount of tubes; formation of horseshoe vortices in front of the tubes; interaction of secondary flows of opposite vorticity at the front and sides of the tubes; recirculating tube wakes; vortices in the wake of the fins; improved air mixing and thinning of boundary layers along with lateral swirl mixing due to the corrugated geometry formed by the wavy fins.

3.1. Overview of Unresolved Issues

In the literature a number of experimental and numerical studies have been performed in order to investigate effects of various geometrical parameters on fluid flow and heat transfer characteristics of plate fin and tube heat exchangers. A considerable amount of experimental and numerical studies were conducted under steady state and laminar conditions. However it is known that most flows encountered have unsteady characteristics with flow separation and recirculation zones for the plate fin and tube heat exchangers. As it is known, the tube diameter based Reynolds number range is typically $100 < Re < 2000$ for the compact heat exchanger surfaces (Shah et al, 1997). However, there is no clear similarity and compatibility between the researchers about a certain Reynolds number value that corresponds to flow unsteadiness as well as flow regime in the heat exchanger passage. Beale and Spalding (1999) observed in their experimental study that, for both inlined and staggered tube banks without fins exhibit **transient** behavior for $Re > 100$. It is obvious that, with the presence of both the tubes and the fins, the flow

will become more complex and 3-D and the Reynolds number corresponding to transient behavior for the plate fin and tube heat exchangers will be much smaller than this value. Kundu et al. (1991a) reported that the flow becomes **unstable** when the Reynolds number is larger than 350 ($Re > 350$) for plate fin and tube heat exchangers. At low Reynolds numbers their experimental data were in good agreement with the numerical results. In their study, the Reynolds number at which the experimental data depart from the numerical results is interpreted as the **transitional Reynolds number**. Similarly, Tsai and Sheu (1998) verified their computational calculations with the experimental data and showed that their computed data becomes increasingly different from the experimental data because of increasing Reynolds numbers continuously approaching the **transitional flow regime**. According to Kundu et al. (1992), the flow attains **turbulent** characteristics when the Reynolds number is larger than 400 ($Re > 400$) based on their pressure measurements and numerical calculations.

Shah et al. (1997) indicated that for compact heat exchangers, there is no clear evidence yet that the flows in the Reynolds number range ($100 < Re < 2000$) are **unsteady laminar** or **low Reynolds number turbulent** flows; although it is possible that gradually the unsteady laminar flows associated with the interruptions become low Reynolds number turbulent flows. Authors indicated that more sophisticated experimentation on small sample fin geometries is required to determine the flow characteristics. According to the authors, the separation and reattachment of the flow and vortices in the wake of the fins affect both heat transfer and pressure drop, and these mechanisms cannot be predicted with **time average modeling**. Kundu et al. (1992) claimed that their experimental data depart from the numerical data because the numerical data for laminar flow do not include the **turbulent mixing** effect. Ciofalo et al. (1996) used several numerical models and claimed that **low Reynolds number turbulent flow model** or **LES** give satisfactory results for determining heat transfer and flow characteristics of plate fin and tube heat exchangers. Shah et al. (1997) indicated that **LES and DNS methods should be further developed** to simulate the local flow and heat transfer characteristics of compact heat exchanger surfaces. Therefore in the present study it is decided to use both space and time

averaged turbulence models besides laminar model in order to simulate the transitional and turbulence effects on the flow for the plate fin and tube heat exchangers.

Similarly, in the literature, most of the numerical studies have been performed using 2-D modeling. However, importance and necessity of 3-D modeling has been reported by some investigators. Zdravistch et al. (1994) performed 2-D simulations for heat transfer predictions in a tube bank without fins. Authors reported the importance of necessity for **3-D simulations** when side wall effects are important, like in tube banks with fins. Therefore it is suggested that in the present study a 3-D transient modeling with the consideration of turbulence effects should be employed in order to capture the flow details and instantaneous informations for predicting the flow and heat transfer characteristics in the heat exchanger flow passage.

3.2. Outline of Dissertation

In recent years, developments in the computer hardware technology and improvements in the speed of personal computers have made it possible to handle fluid flow and heat transfer problems numerically. Numerical modelling has the potential of offering a **flexible and cost effective** means for fluid flow and heat transfer investigations and has the added advantage of reproducing exact geometries, boundary conditions and fluid properties. As indicated by Shah et al. (1997), a comprehensive experimental study of the performance of the compact heat exchanger surfaces is very expensive because of the high cost of tools needed to produce a wide range of geometric variations. Experimental errors that may arise from personel and/or instrumental instabilities and their influence on the results of the study should be also considered. On the other hand, the numerical results should be verified against experimental results to be sure about the accuracy of the numerical solution.

Therefore in the first part of this investigation, the accuracy of the numerical model is tested and validated against the experimental data. In the second part, an

optimization study regarding geometrical characteristics of plate fin and tube heat exchangers have been performed under laminar conditions. In the third part, both laminar and turbulence models are employed in order to show modeling effects on the heat transfer and pressure drop performance of plate fin and tube heat exchangers. In the last section, the best model is selected which is best fitted with the experimental data and the LES method is further improved to give a better solution in terms of heat transfer and fluid flow resolution for the plate fin and tube heat exchanger. Moreover, the time evolution of the flow in the heat exchanger passage is further investigated in order to capture local flow details. In addition, several experimental validations have been performed throughout the investigation in order to show the validity of the present numerical study.

All simulations are performed by using FLUENT finite volume code (FLUENT, 1998). The accuracy of the present numerical model is tested against the PIV experimental data conducted at Mechanical Engineering Department of Cukurova University, in the Fluid Dynamics Research Laboratory.

4. MATERIALS AND METHODS

As it is outlined in Chapter 1 and 2, the need for low weight, space saving and economical heat exchangers have driven the development of compact surfaces. The compact heat exchanger considered in the present investigation employs a compact surface on the air side regarding its fin density. The fin density is expressed in terms of number of fins per meter (fins/m) or number of fins per inch (fins/in). In the present investigation, a series of test simulations have been implemented to obtain an optimum fin density. The test simulations are carried on in order to determine optimum value of tube row number, tube layout (inlined or staggered), fin configuration (plain or wavy), and wavy angle. All of the optimization studies are performed under laminar flow assumption. The simulations are carried out by using the FLUENT finite volume code. After having decided the optimum geometrical parameters of the heat exchanger, effect of turbulence in the heat exchanger passage is investigated by employing various turbulence models available in the FLUENT finite volume code. Finally, the present FLUENT finite volume code is extended to consider time and space variations of turbulence effects. The results are compared with each other and with the available experimental data regarding heat transfer and pressure drop performance of the selected optimum heat exchanger.

In the present study, FIDAP finite element code is used for generation of computational meshes for the plate fin and tube heat exchanger geometries and the FLUENT finite volume code is used for specifying boundary conditions, material properties, basic equations to be solved, setting solution control parameters, calculating the solution and post processing the results. The fluid is considered isothermal, incompressible and Newtonian (viscous) with constant properties and the flow is assumed to be three-dimensional and transient. 3-D Navier-Stokes equations and continuity equations are solved for the solution of the velocity and pressure distribution of the flow field; while the energy equation is solved for the determination of temperature field.

4.1. Governing Equations for Fluid Flow and Heat Transfer

All physical systems obey three fundamental conservation laws: Mass is conserved, Momentum is conserved, and Energy is conserved. The governing equations of fluid flow and heat transfer represent mathematical statements of the conservation laws of physics. The conservation equations are derived by considering the mass, momentum and energy balances for an infinitesimal control volume. Derivation of the mass, the momentum and the energy conservation equations can be found in Appendix A. In the absence of an external body force, differential forms of the governing continuity, momentum and energy equations for the laminar case used in the present study can be given as follows:

1) **Continuity Equation.** The mass conservation or the continuity equation for a time- dependent, three-dimensional, incompressible and Newtonian fluid is given in Eq. (4.1).

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (4.1)$$

2) **Momentum Equation.** The most useful form of the conservation of the momentum equation is obtained by applying Newton's Second Law of motion to an infinitesimal fluid element and re-arranging the viscous stress terms yields the so-called Navier-Stokes Equation.

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_j u_i)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\nu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \quad (4.2)$$

3) **Energy Equation.** The energy of a fluid is defined as the sum of internal (thermal) energy, kinetic energy and gravitational potential energy. The internal energy equation for a time- dependent, three- dimensional, incompressible and Newtonian fluid is given in Eq. (4.3)

$$\frac{\partial T}{\partial t} + \frac{\partial}{\partial x_i}(u_j T) = \alpha \frac{\partial}{\partial x_j} \frac{\partial T}{\partial x_j} \quad (4.3)$$

4.2. Reynolds Averaged Navier Stokes (RANS) Equations of Motion

The Navier- Stokes Equations are a set of non- linear partial differential equations and they can only be able to describe continuous and homogeneous fluid flows approximately. However, at very small scales or under extreme conditions, real fluids made out of mixtures of discrete molecules and other materials, such as suspended particles and dissolved gases, produce different results from the continuous and homogeneous fluids modelled by the Navier- Stokes Equations. Although the full, unsteady Navier- Stokes Equations correctly describe nearly all flows of practical interest, they are too complex for practical solution in many cases and a special “reduced” form of the full equations is often used instead- these are the Reynolds Averaged Navier Stokes (RANS) equations.

In Reynolds Averaging, the solution variables in the instantaneous (exact) Navier-Stokes equations are decomposed into the mean (ensemble-averaged or time-averaged) and fluctuating components. Now all flow variables in the equations can be displaced by sum of a mean (ensemble) and a fluctuating component: Reynolds averaging means replacing the time- varying quantities with time- averaged (mean) and fluctuating components as

$$u = \bar{u} + u'; v = \bar{v} + v'; w = \bar{w} + w'; p = \bar{p} + p'$$

where $\bar{u}, \bar{v}, \bar{w}$ are mean velocity components, and u', v', w' are fluctuating velocity components.

The solution of the full steady Navier- Stokes Equations is sufficiently accurate only for laminar cases of the fluid flow. For turbulent flows the Reynolds averaged form of the equations are most commonly used. The RANS form of the equations introduce new terms that reflect the additional modeling of the small turbulent motions.

In the absence of an external body forces, the governing time averaged conservation equations for an incompressible, Newtonian fluid of constant density and constant viscosity take the following forms:

1) Time averaged continuity equation:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (4.4)$$

2) Time averaged momentum equation:

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial (\bar{u}_j \bar{u}_i)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\nu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \overline{u_i' u_j'} \right] \quad (4.5)$$

3) Time averaged energy equation:

$$\frac{\partial \bar{T}}{\partial t} + \frac{\partial (\bar{u}_j \bar{T})}{\partial x_j} = \alpha \frac{\partial}{\partial x_j} \frac{\partial \bar{T}}{\partial x_j} - \overline{q_j} \quad (4.6)$$

Reynolds averaging introduces additional terms in the momentum and energy equations which are known as “Reynolds (turbulent) stress”, $-\rho \overline{u_i' u_j'}$, and the “Reynolds (turbulent) heat flux”, $-\overline{q_j}$, which appear on the right-hand side of the momentum and the energy equations, respectively. The task of turbulence modeling is to model these unknown, higher-order, extra terms in the mean flow equations and thus close the system of equations. The need to model these correlations is the ‘closure problem’. It is also possible to model a transport equation for the heat flux, but this is not a common practice. Instead, a turbulent thermal diffusivity is defined proportional to the turbulent viscosity. The constant of proportionality is called the turbulent Prandtl number, Pr_t ($0.85 < Pr_t < 0.90$).

4.3. Overview of Turbulence Modeling

For laminar flows the conservation equations can be used to describe the flow. Turbulence however, leads to rapid velocity fluctuations in both space and time. So for turbulent flows, the amount of calculation effort required to capture both the time and space variation of the variables is extremely large. This leads to the concept of turbulence modeling. Most engineering models of turbulent flow assume that the quantities at a given point in space and time can be described as a superposition of some mean quantity, which may vary slowly with time, and a random component which varies rapidly.

Thus, prediction of turbulent flows is achieved by turbulence modeling. The governing equations for turbulent flows are well- known (see Appendix B) and are the non- linear, unsteady, three- dimensional Navier- Stokes Equations.

It is the main task of turbulence modeling to develop computational procedures of sufficient accuracy and generality for engineers to predict the Reynolds stresses, Reynolds heat fluxes and the scalar transport terms in order for closing the Navier- Stokes and the Energy Equations by providing required turbulence information. Therefore many different types of turbulence models have been developed:

1. Models that use the Boussinesq Approximation: These are the eddy viscosity models.
2. Models that solve directly for the Reynolds Stresses: These become complicated fast by introducing further terms requiring modeling.
3. Models not based on time- averaging: These are the Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) methods.

4.4. Turbulence Models

A turbulence model is a computational procedure to close the system of mean flow equations so that a more or less wide variety of flow problems can be calculated. For most engineering purposes it is unnecessary to resolve the details of the turbulent fluctuations. Only the effects of the turbulence on the mean flow are usually sought. The most common turbulence models are classified as follows:

4.4.1. Time Averaged Turbulence Models

Time Averaged Turbulence Models are also referred as RANS Equations Based Turbulence Models or Classical Models. These are:

1. Zero Equation Model (Mixing Length Model)
2. One Equation Model
3. Two Equation Model ($k - \varepsilon$ Model, $k - \omega$ Model)
4. Reynolds Stress Model
5. Algebraic Stress Model

These models model all scales of eddies present in the flow. Zero, One and Two Equation Models use Boussinesq Hypothesis to relate the Reynolds stresses to the time averaged velocity gradients as shown below:

$$-\overline{\rho u'v'} = \mu_t \left(\frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right) \quad (4.7)$$

where μ_t is the turbulent (eddy) viscosity. The advantage of this approach is that relatively low computational cost is required to compute the turbulent viscosity.

4.4.1.1. Zero Equation Model

The Zero Equation Model was the first turbulence model proposed by Prandtl in 1925 and is also known as Prandtl Mixing Length Model. In zero- equation model,

the flow is characterized by one velocity scale and one length scale and the turbulent viscosity is assumed to be constant. Therefore no transport of turbulence is resolved in the zero equation model. The eddy viscosity is expressed as

$$\mu_t = \rho C_\mu L_m^2 \left| \frac{\partial \bar{u}}{\partial y} \right| \quad (4.8)$$

where L_m is the mixing length and C_μ is a constant. The model can be successively applied to the simple flows and the mixing length can be specified by an empirical formula in most situations. However, the model fail for separating flows and cannot be applied to the rapidly developing and recirculating flows where convective or diffusive transport of turbulence are important.

4.4.1.2. One Equation Model

In one- equation model a transport equation for the turbulent kinetic energy is solved and some physically based arguments for the mixing length is solved. This means that the turbulent eddy viscosity now becomes a dependent variable that varies spatially and with time for unsteady flows. The turbulent kinetic energy is considered to be the velocity scale since it is contained in the large scale fluctuations and is expressed as follows:

$$k = \frac{1}{2} (\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) \quad (4.9)$$

The turbulent eddy viscosity is modeled by

$$\mu_t = \rho C_\mu' \sqrt{k} L_s \quad (4.10)$$

where C'_μ and L_s represent empirical constant and length scale, respectively. The length scale distribution cannot be specified empirically for complex flows. Although theoretical formulae for determining the length scale are proposed based on the mean flow, they are very dependent on the problem type, they have not been tested sufficiently and they require rather expensive computing time.

4.4.1.3. Two Equation Models

In two equation models, the RANS equations are closed by assuming the turbulent stresses are proportional to the mean velocity gradients and the constant of proportionality is the turbulent viscosity (Boussinesq Hypothesis). The constant of proportionality is allowed to vary through out the flow field and has been correlated in terms of the turbulent kinetic energy and dissipation rate. Various transport equations are developed for the turbulent kinetic energy and dissipation rate.

(1) Standard Linear Turbulence k - ε Model

The standard linear k - ε turbulence model is well known and the model has been tested for vortex shedding flow by Majumdar and Rodi (1985) previously. Turbulent kinematic viscosity, ν_t , is related to kinetic energy k and turbulent dissipation ε by a following formula:

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad (4.11)$$

The forms of the k and ε equation of the Standard Linear k - ε model are then gives as follows:

$$\frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left(\frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + \nu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - \varepsilon \quad (4.12)$$

$$\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left(\frac{\nu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + C_1 \frac{\varepsilon}{k} \nu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} - C_2 \frac{\varepsilon^2}{k} \quad (4.13)$$

The coefficients C_1 , C_2 , C_μ , σ_k , σ_ε are constants in the sense that they are not changed in any calculation. However, these constants need to be changed in order to accommodate the effects such as low Reynolds number, near wall, etc. These five coefficients have a value as shown below for the Standart $k-\varepsilon$ model:

$$C_\mu = 0.09$$

$$C_1 = 1.44$$

$$C_2 = 1.92$$

$$\sigma_k = 1.00$$

$$\sigma_\varepsilon = 1.30$$

The turbulent heat transport on the other hand is modelled by using the concept of Reynolds' analogy to turbulent momentum transfer by the use of Eq. (4.6).

The standard linear $k-\varepsilon$ turbulence model is extended by Launder and Spalding (1972) and named by $k-\omega$ and $k-L$ models where L_s is the length characterizing the macroscale of turbulence and ω is the time average square of the vorticity fluctuations as defined by Wilcox and Rubesin (1980):

$$\omega = \frac{\varepsilon^2}{(C_f k)^2} \quad (4.14)$$

$$\varepsilon = \nu \frac{\overline{\partial u_j' \partial u_j'}}{\partial x_i \partial x_i} \quad (4.15)$$

$$L_s = \frac{C_f k^{3/2}}{\varepsilon} \quad (4.16)$$

The standard linear and extended k - ε turbulence models are not sufficient for the predictions of near wall flow at high Reynolds numbers.

(2) Non-Linear RNG k - ε Turbulence Model

The Renormalization Group (RNG) k - ε model is proposed by Speziale (1987) and has been validated against internal flow and impinging jet problem by Rabbit (1997) and later is developed by Yakhot et al. (1992). This model is very similar in form to the standard and extended k - ε turbulence models, however the RNG k - ε model differs from the standard model by the inclusion of an additional sink term in the turbulence dissipation equation to account for non-equilibrium strain rates and employs different values for the various model coefficients. The form of the k equation remains same. The turbulence dissipation, ε equation of the RNG k - ε model includes the following sink term

$$\frac{C_\mu \eta^3 (1 - \eta/\eta_0) \varepsilon^2}{1 + \beta \eta^3} \frac{1}{k} \quad (4.17)$$

In the above term the extra term employs the parameter η , which is the ratio between the characteristic time scales of turbulence and mean flow field as follows:

$$\eta = S \frac{k}{\varepsilon}$$

where

$$S = \sqrt{2S_{ij}S_{ij}} = \sqrt{G/\mu_t}$$

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (4.18)$$

The primary model coefficients of the RNG k - ε turbulence model are C_μ , C_1 , C_2 , σ_k , σ_ε and Von Karman constant κ . Recommended values of this model coefficients are as follows:

$$C_\mu = 0.085$$

$$C_1 = 1.41$$

$$C_2 = 1.68$$

$$\sigma_k = 0.7179$$

$$\sigma_\varepsilon = 0.7179$$

$$\kappa = 0.3875$$

(3) Realizable k- ε Turbulence Model

The term “realizable” means that the model satisfies certain mathematical constraints on the normal stress, consistent with the physics of turbulent flows. The realizable k - ε model proposed by Shih et al. (1995) intends to address the shortcomings of traditional k - ε turbulence models by adopting a new eddy-viscosity formula involving a variable C_μ originally proposed by Reynolds (1987) and a new model equation for dissipation based on the dynamic equation of the mean-square vorticity fluctuation. The kinetic energy equation is the same as that in the standard k - ε model and RNG k - ε model, except for model constants. However, the form of the dissipation equation is quite different from those in the standard and RNG-based k - ε models. One of the significant features is that the production term in the dissipation equation does not involve the production of kinetic energy i.e. it does not contain the same G_k term as the other k - ε models. It is believed that the present form better represents the spectral energy transfer.

The modified dissipation equation is given by

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} \quad (4.19)$$

where

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right] \quad (4.20)$$

and

$$\eta = \frac{Sk}{\varepsilon} \quad (4.21)$$

In the above equation C_2 is constant, whereas C_μ is no longer constant but is a function of the time averaged strain and the turbulent fields. C_μ in the eddy viscosity formula in Equation (4.11) can be shown to recover the standard value of 0.09 for an inertial sub-layer in an equilibrium boundary layer. This model has been extensively validated for a wide range of flows including rotating homogenous shear flows, channel and boundary layer flows, and separated flows.

One limitation of the realizable $k-\varepsilon$ model is that it produces non-physical turbulent viscosities in situations when the computational domain contains both rotating and stationary fluid zones (e.g., multiple reference frames, rotating sliding meshes). This is due to the fact that the realizable $k-\varepsilon$ model includes the effects of mean rotation in the definition of the turbulent viscosity

4.4.1.4. Reynolds Stress Model

The alternative approach, embodied in the Reynolds Stress Model, is to solve transport equations for each of the terms in the Reynolds stress tensor. An additional scale-determining equation (normally for ε) is also required. In other words, 6 transport equations for the Reynolds stresses and 1 transport equation for the dissipation rate are solved.

In many cases, models based on the Boussinesq hypothesis perform very well, and the additional computational expense of the Reynolds stress model is not justified. However, the RSM is clearly superior for situations in which the anisotropy of turbulence has a dominant effect on the mean flow. Such cases include highly swirling flows and stress-driven secondary flows.

4.4.1.5. Algebraic Stress Model

Algebraic Stress Models determine the Reynolds stresses as functions of k , ε , and mean velocity gradients i.e., the Reynolds stress model is reduced to a set of algebraic equations. The model predicts most qualitative features and provides fair quantitative for flows with secondary motions.

4.4.2. Space Averaged (Filtered) Turbulence Models

Space averaged (filtered) turbulence models are the ones where the time-dependent flow equations are solved (directly computed) for the mean flow and the largest -and more easily resolvable- eddies; whereas the effects of the smallest eddies are modeled. As it is known, the large eddies interact strongly with the mean flow and contain most of the energy. They are highly responsible for transport of momentum, energy and other scalars, directly affecting the mean fields. Additionally they are anisotropic and strongly dependent on the flow configuration, boundary conditions, and flow parameters. So, they should be simulated directly, because they are difficult to model by an universal approach. The small scale structures on the other hand, produced by stretching of eddies from the larger eddies, are dependent little on the problem and they are more isotropic and consequently more universal. So they are easy to model and they can be described by a universal model. The smallest scales are responsible for the dissipation of turbulence kinetic energy.

LES models are based on space- averaged equations of fluid flow where the Navier-Stokes equations are filtered so that the flow variables are split into resolvable and sub-grid scale motions. LES directly computes (resolves) large eddies

and models only small (unresolvable) eddies within the flow. The chance of finding a universal model is much higher when only small eddies are modeled. Since the small eddies contain less turbulent kinetic energy, they have a universal behavior, and are more isotropic, it is relatively easier to model them. LES requires large computational effort and number of grid points with a great success in unsteady calculations. So this approach results in a good model of the main effects of turbulence.

In LES the small scale motion is extracted from the main flow by filtering. While the main flow field is solved from the Navier- Stokes Equations, the smallest eddies are modeled. The filtering process effectively filters out the eddies whose scales are smaller than the filter width (grid spacing) used in the computations. The resulting equations thus govern the dynamics of large eddies. A filtered variable is defined by

$$\hat{\Phi}(x) = \int_D \Phi(x') G(x, x') dx' \quad (4.22)$$

where D is the fluid domain, and G is the filter function that determines the scale of the resolved eddies. There are many kinds of filter functions that can be used. The filter function used in the present finite volume code is

$$G(x, x') = \begin{cases} 1/V, & x' \in V \\ 0, & x' \text{ otherwise} \end{cases}$$

where V is the volume of a computational cell.

Applying the filtering procedure to the continuity, momentum and energy equations,

$$\frac{\partial \hat{u}_i}{\partial x_i} = 0 \quad (4.23)$$

$$\frac{\partial \hat{u}_i}{\partial t} + \frac{\partial(\hat{u}_j \hat{u}_i)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \hat{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\nu \left(\frac{\partial \hat{u}_i}{\partial x_j} + \frac{\partial \hat{u}_j}{\partial x_i} \right) - \tau_{ij} \right] \quad (4.24)$$

$$\frac{\partial \hat{T}}{\partial t} + \frac{\partial(\hat{u}_j \hat{T})}{\partial x_j} = \alpha \frac{\partial}{\partial x_j} \frac{\partial \hat{u}_i}{\partial x_j} - q_j \quad (4.25)$$

where τ_{ij} and q_j are filtered stress and filtered heat flux, respectively. They are also called subgrid-scale stress and subgrid-scale heat flux which are unknown and require modeling. The majority of the subgrid-scale models in use today are eddy viscosity models of the following form:

$$\tau_{ij} = -\nu_t \left(\frac{\partial \hat{u}_i}{\partial x_j} + \frac{\partial \hat{u}_j}{\partial x_i} \right) \quad (4.26)$$

$$q_j = -\frac{\nu_t}{Pr_t} \frac{\partial \hat{T}}{\partial x_j} \quad (4.27)$$

4.4.2.1. Smagorinsky-Lilly Sub-Grid Scale Model (1963)

The most basic of subgrid-scale models was proposed by Smagorinsky (1963) and further developed by Lilly (1966). In this model, the eddy viscosity is modeled by

$$\mu_t = \rho L_s^2 |\hat{S}| \quad (4.28)$$

where L_s is the mixing length for sub-grid scales and $\hat{S}$ is the strain rate tensor.

$$|\hat{S}| \equiv \sqrt{2\hat{S}_{ij}\hat{S}_{ij}} \quad (4.29)$$

$$\hat{S}_{ij} = \begin{pmatrix} S_{xx} & S_{xy} & S_{xz} \\ S_{yx} & S_{yy} & S_{yz} \\ S_{zx} & S_{zy} & S_{zz} \end{pmatrix} = \frac{1}{2} \left[\frac{\partial \hat{u}_i}{\partial x_j} + \frac{\partial \hat{u}_j}{\partial x_i} \right] \quad (4.30)$$

In the present finite volume code, L_s is computed as

$$L_s = \min(\kappa D, C_s V^{1/3}) \quad (4.31)$$

where $\kappa=0.42$, D is the distance to the closest wall and V is the volume of the computational cell. Lilly (1966) derived a value of 0.23 for C_s from homogenous isotropic turbulence in the inertial sub-range. However this value was found to cause excessive damping of large-scale fluctuations in the presence of mean shear or in transitional flows $C_s=0.10$ is used for all simulations here.

4.4.2.2. RNG Based Sub-Grid Scale Model

Yakhot et al. (1989) applied the RNG theory to derive a model for the sub-grid scale eddy viscosity. The RNG procedure results in an effective sub-grid viscosity, $\mu_{eff} = \mu + \mu_t$, which is given by

$$\mu_{eff} = \mu \left[1 + H \left(\frac{\mu_s^2 \mu_{eff}}{\mu^3} - c \right) \right]^{1/3} \quad (2.32)$$

$$\mu_s = (C_{mg} V^{1/3})^2 \sqrt{2 \hat{S}_{ij} \hat{S}_{ij}} \quad (2.33)$$

and $H(x)$ is Heaviside function,

$$H(x) = \begin{cases} x, & \text{for } x > 0, \\ 0, & \text{for } x \leq 0. \end{cases}$$

The theory gives $C_{mg}=0.157$ and $c=100$. In highly turbulent regions of the flow ($\mu_t \gg \mu$), $\mu_{eff} \cong \mu_s$ and the RNG based sub-grid scale model reduces to the Smagorinsky-Lilly model with a different model constant. In low Reynolds number regions of the flow, the effective viscosity recovers molecular viscosity. This enables the RNG based sub-grid scale eddy viscosity to model low Reynolds number effects encountered in transitional flows near wall regions.

4.4.2.3. Dynamical LES Model: The Yoshizawa TSDIA Model (1984):

The Yoshizawa Two- Scale Direct Interaction Approximation (TSDIA) model (1984) is based on the combination of Kraichnan' s Direct- Interaction Analysis with a technique of separation of scales of a mean or slow (grid scale) field and a fluctuating or fast (subgrid scale) field, the resulting subgrid scale stress is written in terms of the grid scale velocity, the grid scale pressure, and the subgrid scale turbulent kinetic energy. **In this dynamical model, μ_T is expressed the same way as in (4.28), but C_s , as the length scale parameter, is no longer a constant as it is in the Smagorinsky model; it varies with time and space. The coefficient C_s is given as (Yoshizawa, 1982)**

$$C_s = C_{s0}(1 + C_A \Sigma) \quad (4.34)$$

The two model constants are C_{s0} and C_A :

$$C_{s0} = 0.15 \text{ (model constant)}$$

$$C_A = 0.10 \text{ (model constant)}$$

Σ is defined as $D\hat{S}/Dt/\hat{S}^2$ with D/Dt as the material derivative:

$$\Sigma = \frac{D\hat{S}}{\hat{S}^2} \quad (4.35)$$

where

$$\underbrace{\frac{D\hat{S}}{dt}}_{\text{total derivative}} = \underbrace{\frac{\partial \hat{S}}{\partial t}}_{\text{local derivative}} + \underbrace{u \frac{\partial \hat{S}}{\partial x} + v \frac{\partial \hat{S}}{\partial y} + w \frac{\partial \hat{S}}{\partial z}}_{\text{convective derivative}} \quad (4.36)$$

The quantities $\hat{S}$ is the strain rate magnitude and is equal to $(2S_{ij}S_{ij})^{1/2}$, as defined in Eq. (4.29) before, and $\hat{S}_{ij}$ as defined in Eq. (4.30)

in 3-D,

$$\begin{aligned} \hat{S} = & \left(\frac{\partial u}{\partial x} \frac{\partial u}{\partial x} \right) + \left(\frac{\partial v}{\partial y} \frac{\partial v}{\partial y} \right) + \left(\frac{\partial w}{\partial z} \frac{\partial w}{\partial z} \right) + \frac{1}{2} \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 \\ & + \frac{1}{2} \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 + \frac{1}{2} \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)^2 \end{aligned} \quad (4.37)$$

The term Σ , in the Yoshizawa's model, brings the influence of the resolvable field, including the time rate of change and the advection effect of the strain rate S in to the calculation of C_s , which may be viewed as a length scale parameter. This feature is absent in the classical Smagorinsky model where C_s is the prescribed constant.

The FLUENT Finite Volume Code used in the present investigation does not include the Dynamic LES Method. Therefore, a new program is developed written in the C language, compiled and linked to the code through a user defined function to calculate the value of the sub-grid scale viscosity within a proper subroutine and then sent this value to the main program (Appendix C).

4.4.3. Direct Numerical Simulation

It is theoretically possible to directly resolve the whole spectrum of turbulent scales using an approach known as Direct Numerical Simulation (DNS). When the DNS is applied, the computational domain must be large enough and the computational grid dense enough. Otherwise all features of turbulence will not be described correctly. The grid must be able to capture also the smallest scales of motion. Therefore, DNS requires a huge amount of computer capacity and high order numerical methods to simulate all scales of motion present in the flow. Generally, scientists or engineers are more interested in the description of the large scales of the flow, which often contain the desired information about turbulent transfers of momentum or heat for example: it is these large scales which will be simulated on the computer. Therefore DNS is not feasible for practical engineering problems due to its large computational cost requirement. As it is also stated by Lesieur (1997), for a weakly viscous fluid, it is not possible in the near future (and perhaps not in the distant future either) to simulate explicitly all the scales of motion from the smallest to the largest.

The number of grid points required for a DNS is determined by the ratio of the respective length scales of the energy carrying and dissipative eddies. Because of the disparity between the diffusive and dissipative scales is a function of the Reynolds number, the application of DNS is limited to flows at moderate Reynolds numbers.

4.5. Near Wall Treatment

Turbulent flows are significantly affected by the presence of walls. The mean velocity field is affected through the no-slip condition that has to be satisfied at the wall. The turbulence is also changed by the presence of the wall. Very close to the wall, viscous damping reduces the tangential velocity fluctuations, while kinematic blocking reduces the normal fluctuations. Toward the outer part of the near wall region, the turbulence is rapidly augmented by the production of turbulent kinetic

energy due to the large gradients in mean velocity. Accurate representation of the flow in the near wall region determines successful predictions of wall- bounded turbulent flows (FLUENT, 1998).

Numerous experiments have shown that the near-wall region can be largely subdivided into three layers. In the innermost layer, called the “viscous sublayer”, the flow is almost laminar, and the (molecular) viscosity plays a dominant role in momentum and heat or mass transfer. In the outer layer, called the fully-turbulent layer, turbulence plays a major role. Finally, there is an interim region between the viscous sublayer and the fully turbulent layer where the effects of molecular viscosity and turbulence are equally important. Figure 4.1 illustrates these subdivisions of the near-wall region, plotted in semi-log coordinates.

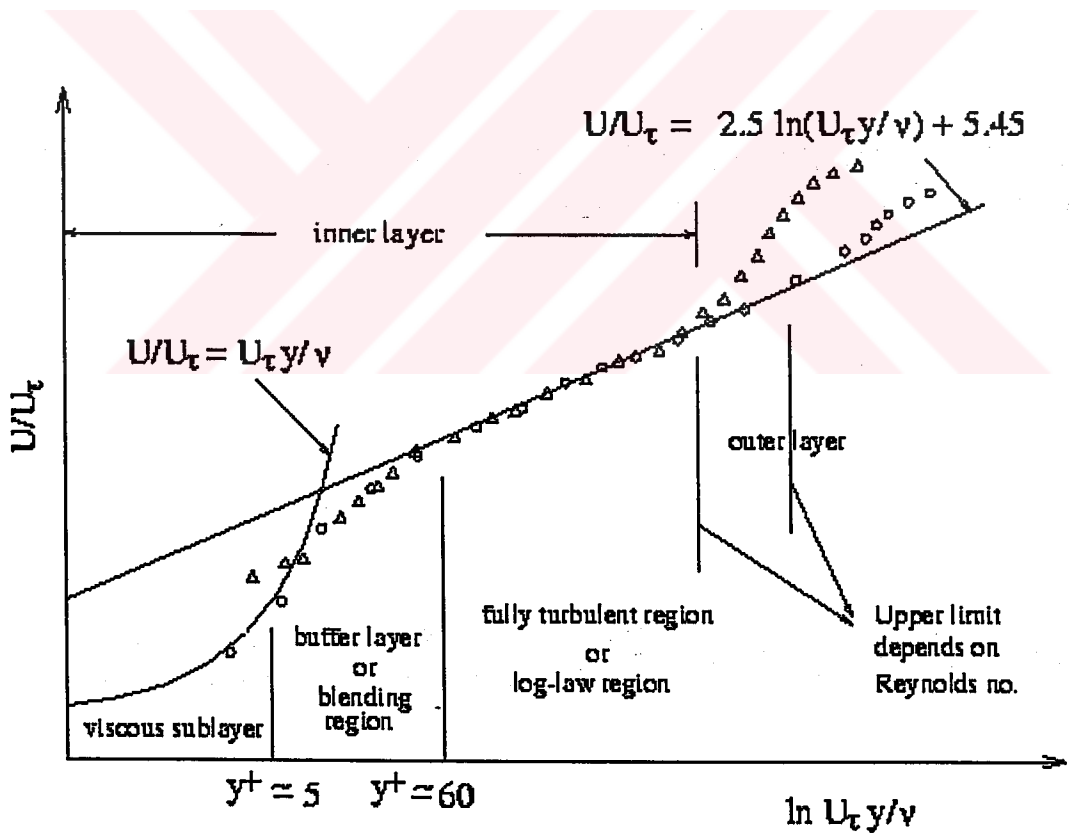


Figure 4.1 Subdivisions of near wall region (FLUENT, 1998)

Traditionally, are two approaches for modeling the near-wall region. In one approach, the viscosity-affected inner region (viscous sublayer and buffer layer) is not resolved. Instead, semi-empirical formulas called “wall functions” are used to bridge the viscosity-affected region between the wall and the fully-turbulent region. The use of wall functions obviates the need to modify the turbulence models to account for the presence of the wall. In most high-Reynolds-number flows, the wall function approach substantially saves computational resources, because the viscosity-affected near-wall region, in which the solution variables change most rapidly, does not need to be resolved. The wall function approach is popular because it is economical, robust, and reasonably accurate. The wall function approach, however, is inadequate in situations where the low-Reynolds-number effects are pervasive in the flow domain in question, and the hypotheses underlying the wall functions cease to be valid. Such situations require near-wall models that are valid in the viscosity-affected region and accordingly integrable all the way to the wall (FLUENT, 1998).

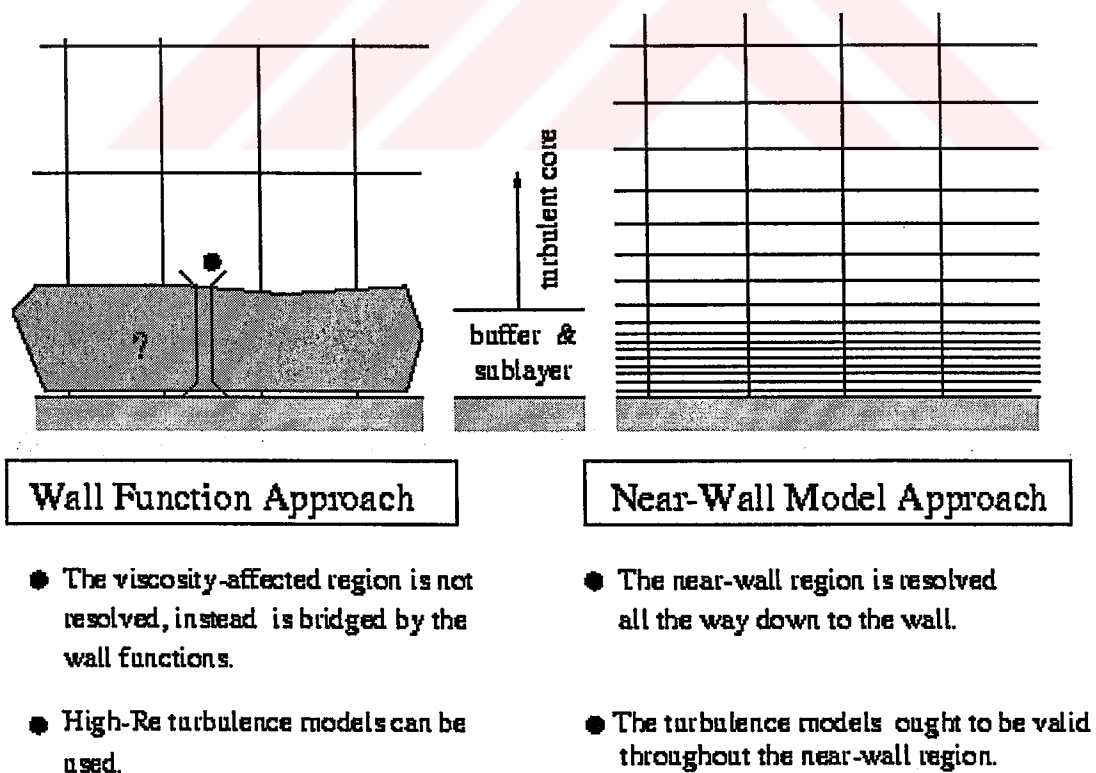


Figure 4.2. Near- Wall Treatments (FLUENT, 1998)

In the present study, all wall functions available in the FLUENT finite volume code are tested and compared against the experimental data and the best wall model is chosen as Enhanced Wall Treatment as discussed in detail in Section 5.4.

4.6. Comparison of Different Turbulence Models

The simulation of engineering turbulent flows is restricted to turbulence-modeling practices based on Reynolds- Averaged Navier- Stokes (RANS) Equations, Large Eddy Simulation (LES), and Direct Numerical Simulation (DNS). In the RANS approach, all the turbulent motions are modeled, resulting in a significant savings in computational effort. As opposed to the RANS approach, in LES, the largest scales of turbulence are simulated (resolved directly) while the effect of the discarded small scales are modeled. At the same time, unlike RANS, LES simulations retain the full three- dimensionality and time dependence of the fluctuating turbulent field. The primary advantage of LES is the reduced influence of the turbulence model, which significantly increases the computational effort in comparison to RANS (Lübcke et al., 2001). In RANS the turbulent model damps out the smallest scales of motion no matter how dense a grid is used, while in LES the grid density (or filter width) defines the amount of damping. This means that as we use a denser grid in LES the turbulence model includes less damping and we approach direct numerical simulation. One essential consequence is that with LES we obtain a three dimensional flow field for two dimensional geometries, which never happens with RANS. LES is situated somewhere between DNS and the RANS approach.

DNS is the most accurate way to study numerically a turbulent flow. In this approach the flow field is solved directly from the N-S equations and no averaging or turbulence modeling is applied. Thus only the numerical methods affect the accuracy of the solution. From some point of view DNS is the most straightforward approach to CFD. As a drawback DNS requires a huge amount of computer capacity and high order numerical methods (Brandt, 2004).

The application of DNS is limited to flows at moderate Reynolds numbers. In LES approach, the severe Reynolds number restrictions of DNS are bypassed by directly simulating the large scales only and supplying the effect of the missing small scales by a so- called subgrid model. Solving only for the large eddies and modeling the smaller scales results in mesh resolution requirements that are much less restrictive than with DNS. Typically, in the LES, mesh sizes can be at least one order of magnitude smaller than with DNS. Furthermore, the time step sizes will be proportional to the eddy-turnover time, which is much less restrictive than with DNS.

Within the past two decades DNS and LES have made major contributions to the improved understanding of the dynamics and the structure of turbulent flows. DNS and LES provide very detailed information about the full three- dimensional and time- dependent evolution of the flow. (Peyret, 1996). LES is expected to play a major role in the analysis and prediction of complex turbulent flows in the future (Domaradzki and Rogallo, 1990).

4.7. Overview of Computational Fluid Dynamics and Computational Heat Transfer

The physical aspects of any fluid flow and heat transfer phenomenon are governed by three fundamental principles: Mass is conserved; Newton's second law and Energy is conserved. These fundamental principles can be expressed in terms of mathematical equations in the form of integral or partial differential equations. Computational Fluid Dynamics (CFD) and Computational Heat Transfer (CHT) are the science of replacing the integrals or partial derivatives in the conservation equations with discretized algebraic forms, to obtain numbers for the flow field values at discrete points in time and/or space and to obtain a numerical description of the complete flow field of interest.

Of course the instrument which has allowed the practical of CFD and CHT is the high speed digital computer. CFD and CHT solutions generally require the repetitive manipulation of many thousands, even millions, of numbers, a task that is humanly impossible without the aid of a computer. Therefore, advances in both CFD

and CHT and its applications to problems of more and more detail and sophistication, are intimately related to advances in computer hardware, particularly in regard to storage and execution speed. This is why the strongest force driving the development of new computers is coming from the CFD community (Anderson, 1995).

The growing need to optimize thermal processes has made it imperative to numerically simulate the relevant transport phenomena, since experimentation is usually too involved and expensive. However, available analytical and experimental results are of considerable importance in checking the accuracy and validity of numerical results (Jaluria and Torrance, 1986).

4.8. Numerical Method

In order to perform a computer simulation of a physical flow problem, the first step is to build a model of the flow domain. The computational meshes of the heat exchanger models are created by using FIDAP commercial computer code. All simulations are carried out by using FLUENT Finite Volume Code (1998). FLUENT is a computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language.

FLUENT solves the governing integral equations for the conservation of mass, momentum, energy and for other scalars such as turbulence and chemical species by using a finite- volume- based technique. In the finite volume method the solution domain is subdivided into a finite number of discrete continuous control volumes (CVs) or cells, and the conservation equations are applied to each CV. The governing conservation equations are integrated on the individual control volumes to construct the algebraic equations for the discrete dependent variables (unknowns) such as velocity, pressure, temperature and scalars. The discretized equations are linearized and the resultant linear system of equations are solved to yield updated values of the dependent variables. All fluid properties- including density, viscosity, specific heat and conductivity are assumed constant.

4.9. Heat Exchanger Model

The heat exchanger geometry shown in Fig. 1.3 is used to predict the flow and heat transfer properties in the flow passage. The computational domain for the heat exchanger model is shown in Fig. 4.3.

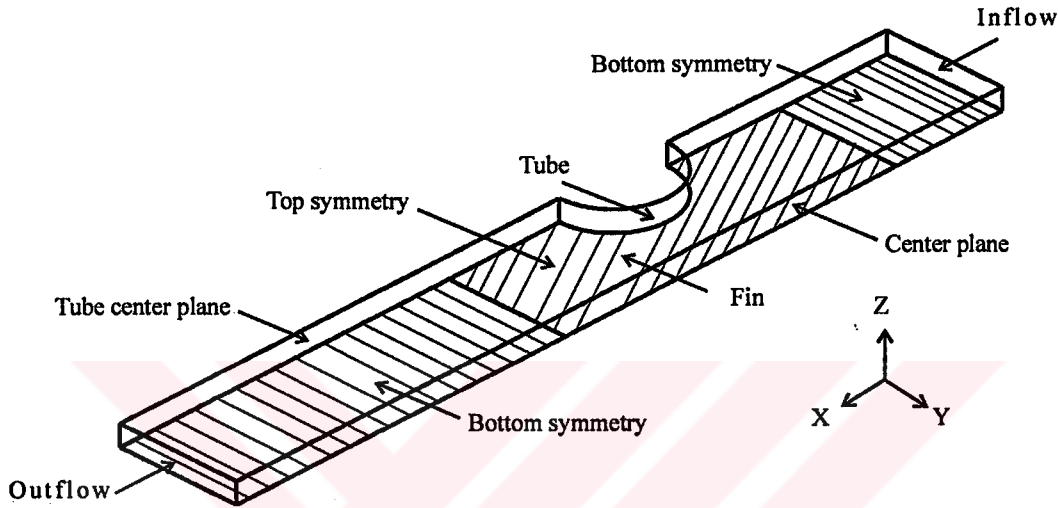


Figure 4.3. Computational domain for the heat exchanger model

The heat exchanger model shown in Fig. 4.3 is symmetric about four planes:

- 1) xz plane through the tube axis (tube center plane)
- 2) xz plane midway between two adjacent tubes (center plane)
- 3) xy plane through the midplane of a fin (top symmetry)
- 4) xy plane midway between two adjacent fins (bottom symmetry)

For computational efficiency, the problem is solved for the region within these four planes. The computational domain extends further both upstream and downstream of the fin plate to allow for the flow entrance and exit development, to accurately accommodate the boundary conditions in the computational domain and to reduce the numerical oscillations. Both plates and tubes are considered to have zero thickness. The flow is in the X -direction.

A test case for a single row configuration has been initially performed to understand the local flow and heat transfer characteristics of the plate fin and tube

heat exchangers. Following the numerical calculations and their validations against experimental data for a single row configuration, the present numerical model has been extended to simulate multi row configurations.

For the single row configuration, the computational domain is defined by $0 \leq x \leq 7.1D$, $0 \leq y \leq 1.06D$, and $0 \leq z \leq F_s/2$ where D is the tube diameter and F_s is the distance between fins. All lengths are non-dimensionalized by the tube diameter. The heat exchanger is itself in $D \leq x \leq 4.05D$ for the single row configuration. The computational domain for the final 4-row configuration is defined by $0 \leq x \leq 30.34D$, $0 \leq y \leq 0.16D$, and $0 \leq z \leq F_s/2$. The heat exchanger is itself in $3.72D \leq x \leq 19.85D$ for this final configuration.

4.10. Computational Mesh

The 3-D finite element mesh is built by using FIDAP computer code (FLUENT, 1998). In order to build the models, firstly the geometry of the flow domain is described by defining discrete set of points (nodes) and collection of these points (elements). The points are joined to create lines, arcs, circles, and finally surfaces. The 3-D geometry of the domain is decomposed into 2-D sub-surfaces. These surfaces are entitled as center plane, tube center plane, top symmetry, bottom symmetry, inflow, outflow, as shown in Fig. 4.3. Secondly the finite element mesh is generated on each surface and throughout the flow domain. The mesh generation means sub-division of the domain into a number of smaller, non-overlapping sub-domains: a grid (or mesh) of cells (or control volumes).

The 3-D unstructured mesh configuration containing non-uniformly grid points for various configurations are illustrated in Fig. 4.4 to Fig. 4.13. The three dimensional finite element mesh with eight noded brick elements are used to mesh the whole computational domain while four-noded quadrilateral control volume elements are used to mesh the surfaces. The critical regions require attention in the development of the mesh. Therefore at the fin and the tube wall surfaces and in regions close to these surfaces a very fine mesh is used in order to accurately resolve

the pressure and velocity gradients as well as secondary flows occurred adjacent to those regions. In the entrance region up to the fin wall, gradients in the y - direction is expected to be very small and flow nominally two dimensional in any xz plane. The velocity gradients upstream and downstream of the fin wall and away from the tube wall are small and relatively a coarse mesh may be used for computation efficiency. However the increment of the mesh between fine and coarse regions should not be change sharply in order to obtain sufficient solution. That' s why a slightly increasing mesh density is used in those regions as illustrated in Figure 4.2 and 4.3.

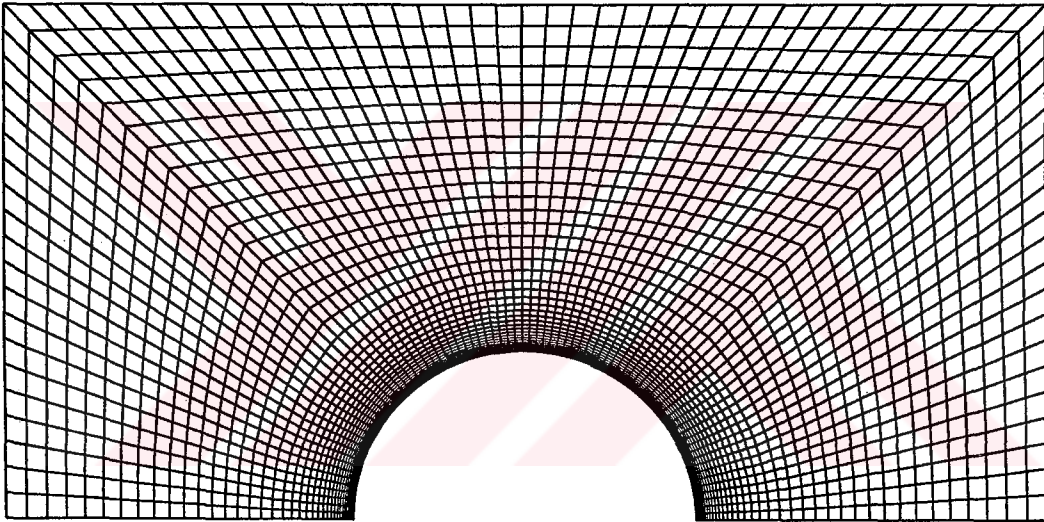


Figure 4.4. Detail of mesh on the fin surface

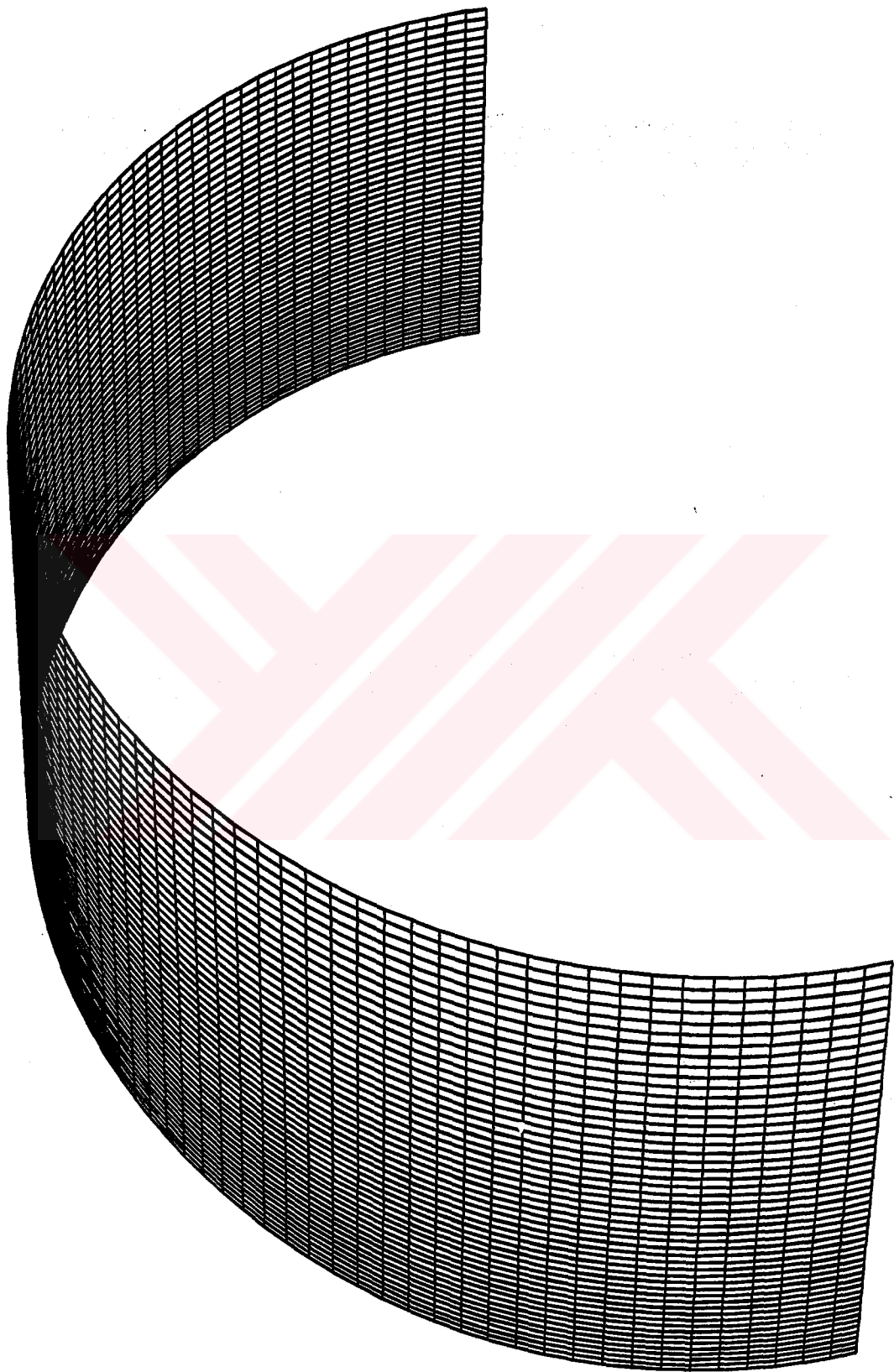


Figure 4.5. Detail of mesh on the tube surface

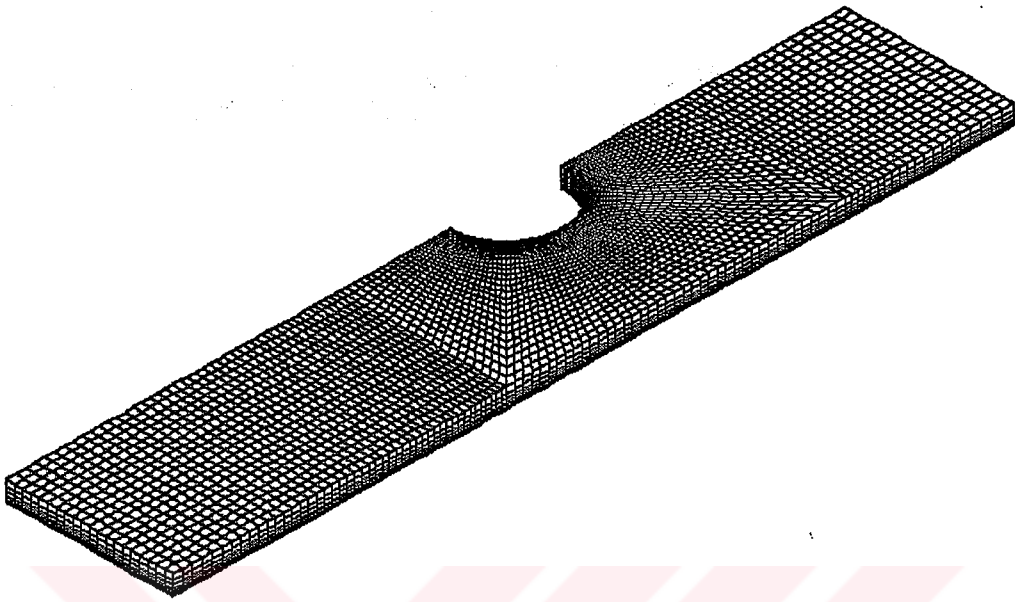


Figure 4.6. Global mesh of the computational domain for the single cylinder configuration

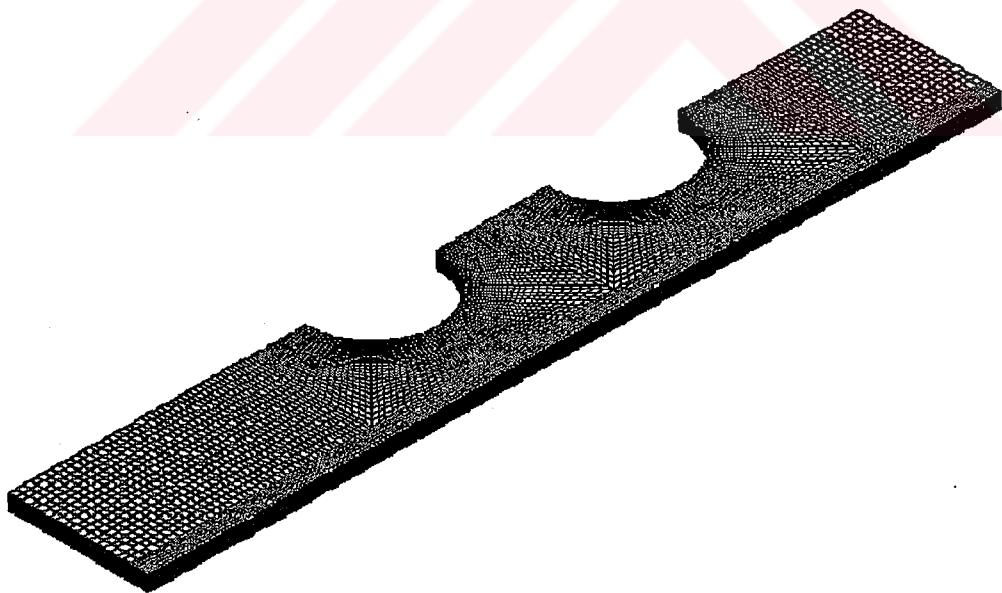


Figure 4.7. Global mesh of the computational domain for the two cylinder of inlined layout

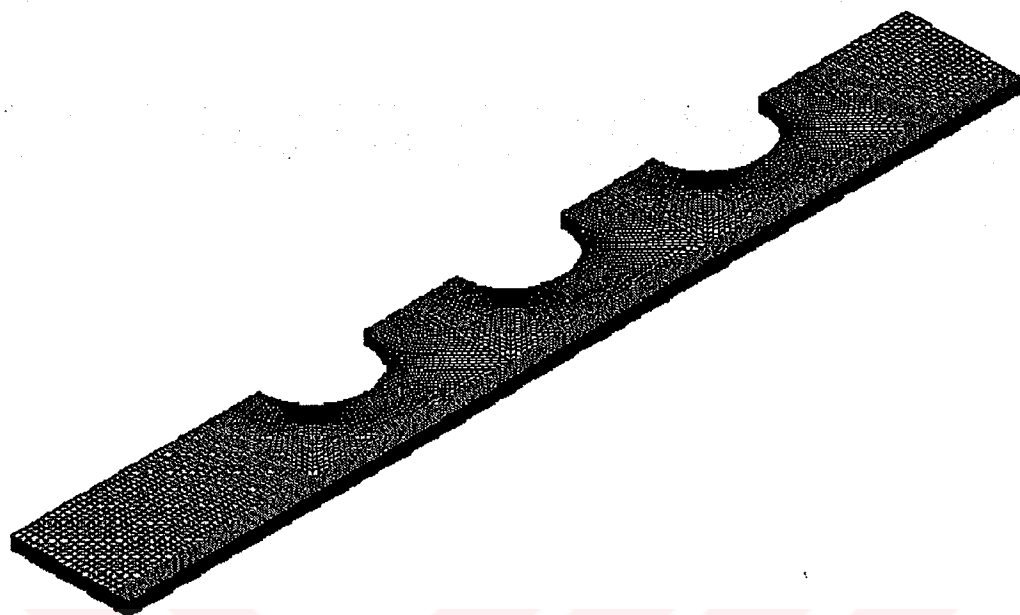


Figure 4.8. Global mesh of the computational domain for the three cylinder of inlined layout

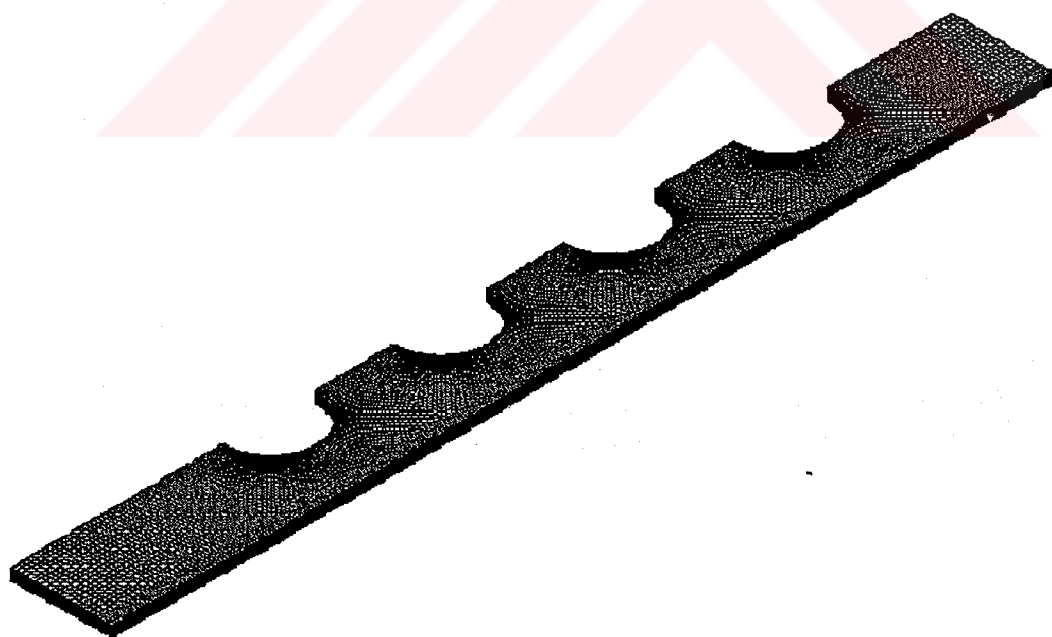


Figure 4.9. Global mesh of the computational domain for the four cylinder of inlined layout

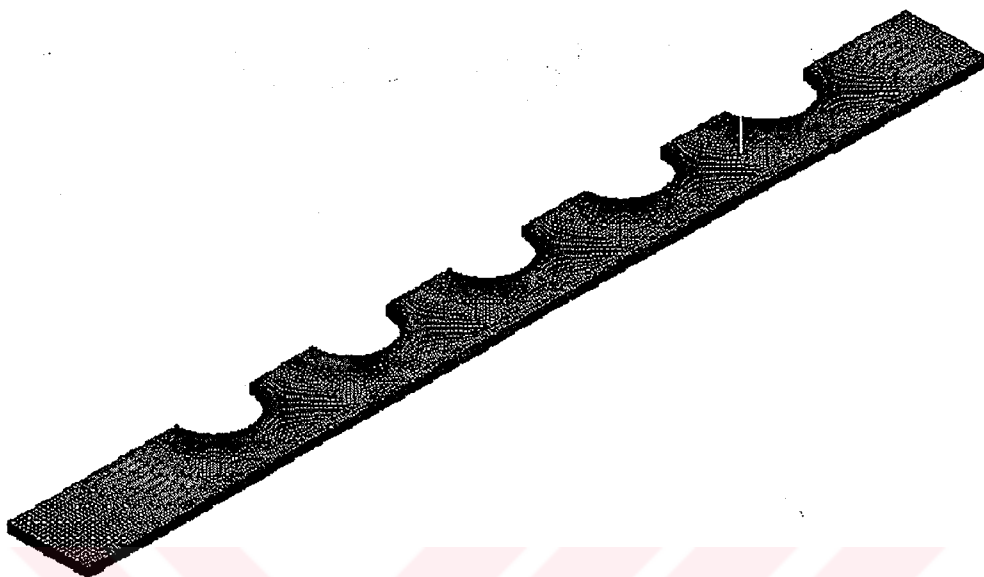


Figure 4.10. Global mesh of the computational domain for the five cylinder of inlined layout

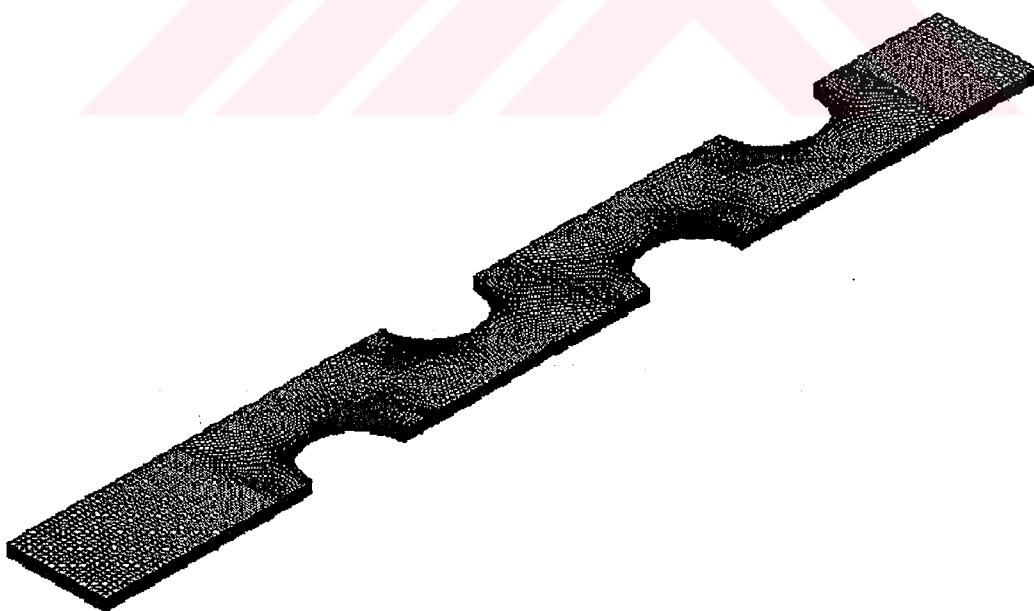


Figure 4.11. Global mesh of the computational domain for the four cylinder of staggered layout

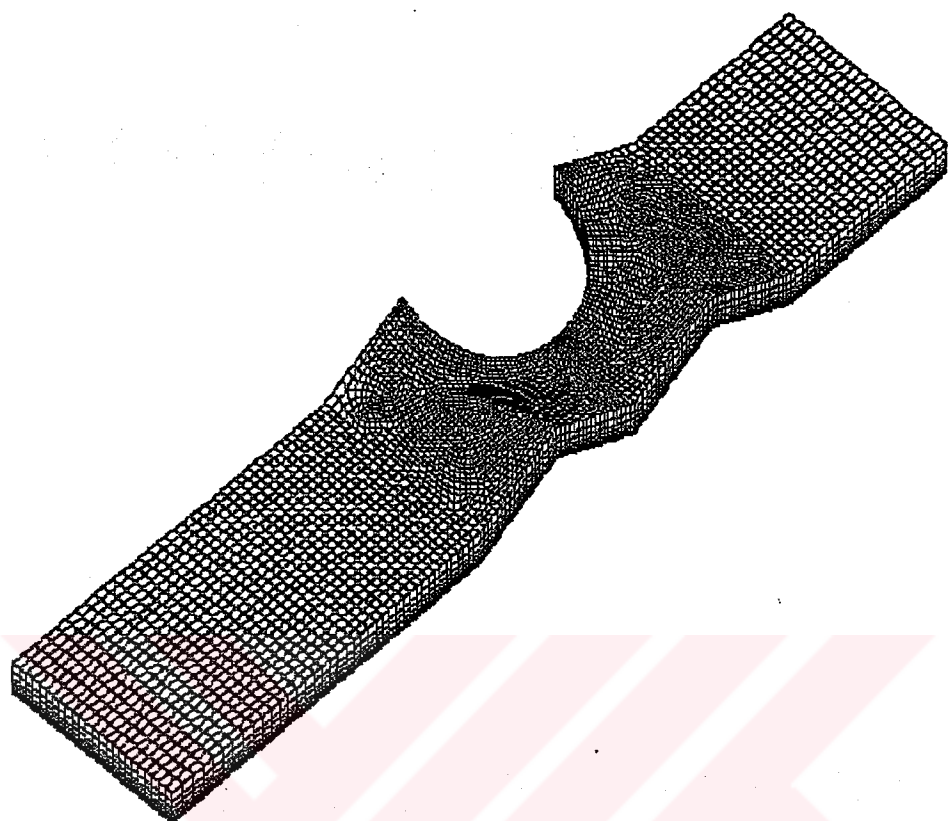


Figure 4.12. Global mesh of the computational domain for the single cylinder of wavy fin configuration



Figure 4.13. Global mesh of the computational domain for the four cylinder of staggered layout and wavy fin configuration

4.11. Boundary Conditions

The boundaries of the computational domain include inlet and outlet boundaries, symmetry planes, solid walls, interior and fluid zones. The several zones are used to assign boundary conditions (Fig. 4.14). All simulations are carried out by considering the following velocity and temperature boundary conditions:

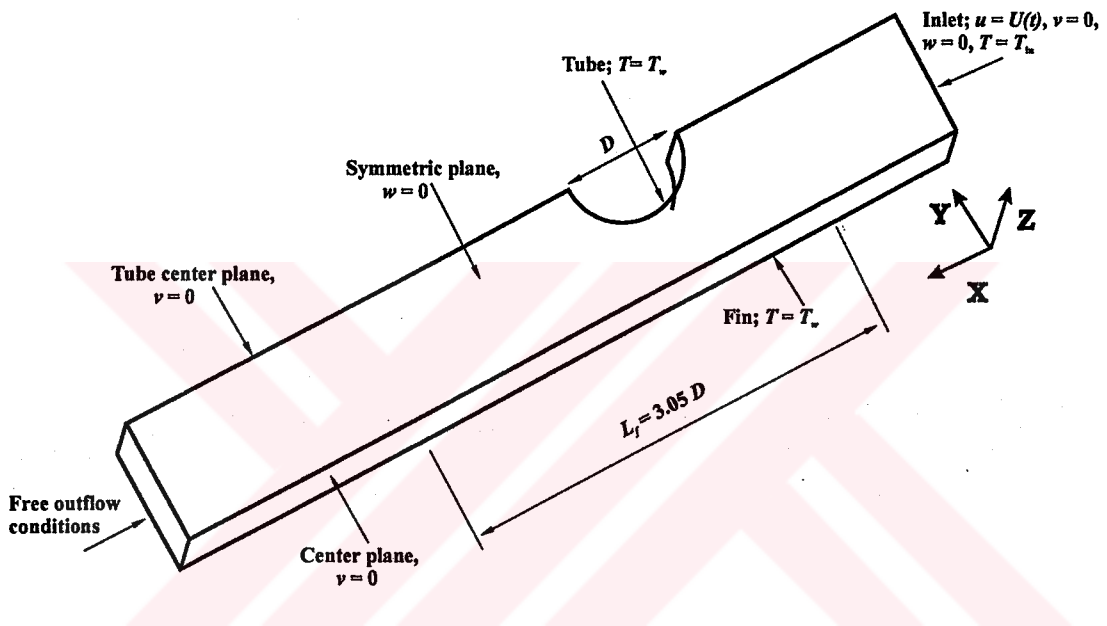


Figure 4.14. Boundary conditions

4.11.1. Boundary Conditions for Steady State Simulations

1. **Velocity Inlet** boundary conditions are used at the inflow section to define the velocity and temperature of the flow at inlet boundaries. For steady state simulations, it is assumed that only a constant and uniform streamwise-direction velocity component exists and other velocity components are assumed to be zero. The value of the constant streamwise- direction velocity component is set as in the experimental studies regarding present day air conditioning machines. The inlet velocity varies between $0.1 \text{ m/s} < u < 2.5 \text{ m/s}$ in the present investigation for different simulations.

2. **Free outflow** boundary condition: The outflow boundary condition is used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem so that mass balance correction is applied at the outflow boundary and other data at the exit plane is extrapolated from the interior. Diffusion fluxes for all flow variables in the direction normal to the exit plane are assumed to be zero. Therefore the outflow velocity is consistent with a -close to- fully developed flow assumption.
3. **Wall** boundary condition is used to bound fluid and solid regions at the fin and tube surfaces: The boundary conditions are no- slip ($u = v = w = 0$) and constant temperature at the solid walls. Wall temperatures are set to a value of 65 °C in case of the single row configuration and 60 °C in case of the multi row configuration to satisfy the experimental conditions. Fin and tube materials are aluminium and copper respectively, regarding the same fin and tube materials used in the present day air- conditioning machines.
4. **Symmetry** boundary conditions at center plane, tube center plane, top symmetry and bottom symmetry: Symmetry zones are the repeating boundaries used to reduce computational effort. Therefore Symmetry boundary conditions are used when the physical geometry of interest have mirror symmetry. Zero normal velocity and zero normal gradients of all variables are assumed which determine a zero flux of all quantities across the symmetry planes, velocity and temperature symmetry at the $y = 0$ and $z = F_s/2$ planes.
5. **Fluid** boundary condition at the fluid zone. Fluid zone is defined as group of cells for which all active equations are solved. The fluid material is set as air.

The fluid Prandtl number is set to a value of 0.72 for the single row case and 0.736 for the multi row case to satisfy experimental conditions in the present numerical study.

4.11.2. Boundary Conditions for Transient Simulations

For transient simulations, a transient velocity inlet and constant temperature boundary conditions at the inflow section is applied: It is assumed that only a streamwise-direction velocity component exists with a transient sinusoidal velocity profile which has the form

$$u = U + A\sin(w_c t) \quad (4.38)$$

to trigger the flow unsteadiness in the flow passage. In Eq. (4.38), the inlet velocity U is taken as 0.122 m/s for the single row heat exchanger configurations and 2.5 m/s or 2 m/s for the multi row configurations depending on the experimental conditions. In all simulations, the amplitude A is taken as one tenth of the inlet velocity. The frequency w_c is equal to the $w_c = 2\pi/T$, in which T is the period. The period is chosen as $T = 0.01$ for the single row case where the inlet velocity is smaller comparing to the multi row case and as $T = 0.1$ for the multi row case where the velocity is larger. Figure 4.15 shows time history of the transient velocity at the inflow section of the computational domain obtained during the simulations for the multi row case where the inlet velocity is equal to 2.5 m/s. Other velocity components are assumed to be zero ($v = w = 0$). Also a constant temperature of 20 °C is set at the flow inlet to satisfy room air conditions. All other boundary conditions are the same as described for the steady state simulations.

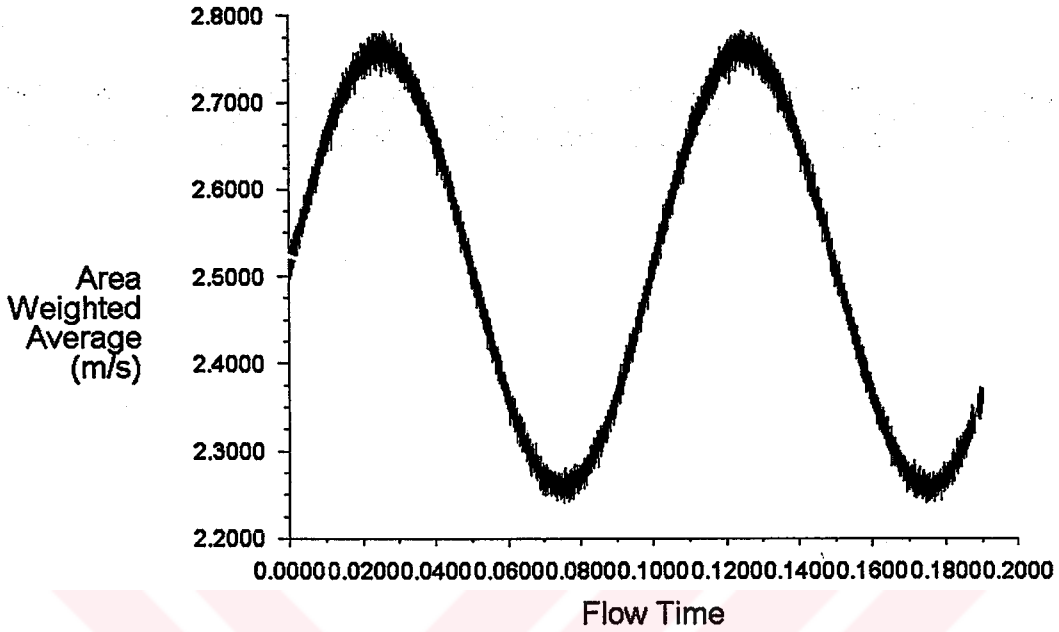


Figure 4.15. Time history of the velocity profile at the inflow section of the computational domain

4.11.3. Boundary Conditions for Turbulence Model Simulations

When the flow enters the domain at the inlet, the transported turbulence quantities should be specified. Therefore, in addition to the above-mentioned velocity and temperature boundary conditions, including a transient velocity inlet profile boundary condition, the turbulent flow properties such as turbulence kinetic energy k and turbulence dissipation rate ε in case of the RANS equations based turbulence model calculations and the turbulence intensity I in case of LES calculations are imposed at the inflow boundary of the computational domain in the Velocity Inlet panel in order to generate the required turbulence at the inlet. In the present study, the inflow turbulence for the LES calculations is generated with the use of random fluctuations superimposed on the time averaged streamwise velocity components at the inlet and the turbulence intensity determines the magnitude of these fluctuations. The turbulence intensity is defined as the ratio of the root mean

square of the velocity fluctuations, u' , to the mean flow velocity, u_{av} . The turbulence intensity at the core of a fully- developed duct flow can be estimated from the following formula derived from an empirical correlation for pipe flows (FLUENT users Manual, 1998).

$$I = \frac{u'}{u_{avg}} = 0.16(Re_D)^{-1/8} \quad (4.39)$$

where Re_D is the Reynolds number based on the tube diameter. Because of the lack of experimental studies regarding turbulent flows in these type of heat exchanger passages, in the present study, the turbulence intensity I in case of LES simulations is calculated by using Eq. (4.39) for different Reynolds numbers and the obtained value of I is imposed at the inflow boundary in the corresponding simulation.

For the RANS equations based turbulence model simulations, k (m^2/s^2) and ε (m^2/s^3) are calculated by using the formulas:

$$k = \frac{3}{2}(u_{av}I)^2 \quad (4.40)$$

$$\varepsilon = C_\mu^{3/4} \frac{k^{3/2}}{l} \quad (4.41)$$

where u_{av} is the average velocity at the inflow section, I is the turbulence intensity as calculated by using Eq. (4.39), $C_\mu=0.09$, l is the characteristic flow length, or the turbulence length scale. The turbulence length scale is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows. An approximate relationship between l and the physical size of the duct is

$$l = 0.07L \quad (4.42)$$

where L is the relevant dimension of the duct. The factor of 0.07 is based on the maximum value of the mixing length in fully-developed turbulent pipe flow, where L is the diameter of the pipe. In a channel of noncircular cross-section, L can be based on the hydraulic diameter. In the present investigation, L is taken as the fin spacing.

In case of turbulence model simulations, the flow is assumed to be turbulent for a given range of Reynolds number and ν is replaced by $(\nu + \nu_t)$ in the governing momentum equation. Similarly, the term $\frac{\nu}{Pr}$ is replaced by

$$\left[\frac{\nu}{Pr} + \frac{\nu_t}{Pr_t} \right]$$

in the energy equation where ν_t is the turbulent kinematic viscosity and Pr_t is the turbulent Prandtl number.

4.12. Computational Details

In the present investigation, in all simulations a near wall methodology that combines the standard wall functions with van Driest (1956) mixing length approach is used. The standard wall functions comprises “law of the wall” for mean velocity and temperature. The viscosity affected inner wall regions are not resolved. Instead, semi empirical formulas called “wall functions” are used to bridge the viscosity affected region between the surface of the tube and the fin and the fully turbulent region. The standard wall functions for the present study suggests the use of logarithmic law for the mean velocity in the viscosity affected inner layer. Reynolds’s analogy between momentum and energy transport also gives a similar logarithmic law for the mean temperature. As in the law of the wall for mean velocity, the law of the wall for temperature comprises logarithmic law for the turbulent region where the effects of turbulence dominate heat conduction.

In addition to the standard calculation procedure, two new user defined functions are written in the C language, compiled and linked to the FLUENT Finite Volume Code to improve the capability of the numerical models used. One is about a sinusoidal, time dependent inlet velocity profile of the form $u = U + A \sin(\omega_c t)$ as described in the specification of transient boundary conditions. The second is about the dynamical LES model that proposed in the previous sections that does not contained in the present FLUENT finite volume code as a Turbulence Method. In case of sinusoidal velocity inlet profile, DEFINE_PROFILE macro and in case of dynamic model, DEFINE_TURBULENT_VISCOSITY macro are used to impose the required properties. They are linked to the FLUENT code through User Defined Functions (UDF). Both UDF's are included in the Appendix C.

4.12.1. Discretization

FLUENT uses a control volume based technique to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control-volume basis. Discretization of the governing equations can be illustrated most easily by considering the steady-state conservation equation for transport of a scalar quantity Φ . This is demonstrated by the following equation written in integral form for an arbitrary control volume V as follows:

$$\oint \rho \vec{v} \cdot d\vec{A} = \oint \Gamma_\phi \nabla \Phi \cdot d\vec{A} + \int_V S_\phi dV \quad (4.43)$$

where ρ is the density, $\vec{v}$ is the velocity vector ($= u\hat{i} + v\hat{j} + w\hat{k}$), $\vec{A}$ is the surface area vector, Γ_ϕ is the diffusion coefficient for Φ , $\nabla \Phi$ is the gradient of Φ ($= \left(\frac{\partial \Phi}{\partial x} \right) \hat{i} + \left(\frac{\partial \Phi}{\partial y} \right) \hat{j} + \left(\frac{\partial \Phi}{\partial z} \right) \hat{k}$), and S_ϕ is the source of Φ per unit volume.

Equation (4.6) is applied to each control volume, or cell, in the computational domain. Discretization of Equation (4.44) on a given cell yields

$$\sum_f^{N_{faces}} \rho_f \vec{v}_f \Phi_f \cdot \vec{A}_f = \sum_f^{N_{faces}} \Gamma_\phi (\nabla \Phi)_n \cdot \vec{A}_f + S_\phi V \quad (4.44)$$

where N_{faces} is the number of faces enclosing cell, Φ_f is the value of Φ convected through face f , $\rho_f \vec{v}_f \cdot \vec{A}_f$ is the mass flux through the face, $\vec{A}_f$ is the area of f , $|\vec{A}| = |A_x \hat{i} + A_y \hat{j} + A_z \hat{k}|$, $(\nabla \Phi)_n$ is the magnitude of $\nabla \Phi$ normal to face f , and V is the cell volume.

Fluent stores discrete values of Φ at the cell centers. However, face values of Φ_f are required for the convection terms in Equation (4.43) and must be interpolated from the cell center values. This is accomplished using an upwind scheme.

4.12.1.1. Second Order Upwind Scheme

Upwinding means that the face value Φ_f is derived from quantities in the cell upstream, or “upwind”, relative to the direction of the normal velocity v_n in Equation (4.7). FLUENT allows to choose several upwind schemes: first-order upwind, second-order upwind, power law, and QUICK. In all simulations, Second order upwind scheme is used for higher order accuracy to be achieved. When second order upwind scheme is used, quantities at cell faces are computed using a multidimensional linear reconstruction approach. In this approach, higher order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about the cell centroid. Thus when second-order upwinding is selected, the face value Φ_f is computed using the following expression:

$$\Phi_f = \Phi + \nabla \Phi \cdot \vec{\Delta s} \quad (4.45)$$

where Φ and $\nabla\Phi$ are the cell- centered value and its gradient in the upstream cell, and $\Delta\vec{s}$ is the displacement vector from the upstream cell centroid to the face centroid. This formulation requires the determination of $\nabla\Phi$ in each cell. This gradient is computed using the divergence theorem, which in discrete form is written as

$$\nabla\Phi = \frac{1}{V} \sum_f^{N_{faces}} \tilde{\Phi}_f \vec{A} \quad (4.46)$$

Here the face values $\tilde{\Phi}_f$ are computed by averaging Φ from the two cells adjacent to the face. Finally, the gradient $\nabla\Phi$ is limited so that no new maxima or minima are introduced.

4.12.1.2. Under Relaxation

Because of the nonlinearity of the equation set being solved by FLUENT, it is necessary to control the value of Φ . This is typically achieved by under- relaxation, which reduces the change of Φ produced during each iteration. In a simple form, the new value of the variable Φ within a cell depends upon the old value, Φ_{old} , the computed change in Φ , $\Delta\Phi$, and the under relaxation factor, α , as follows:

$$\Phi = \Phi_{old} + \alpha\Delta\Phi \quad (4.47)$$

In the present study, in all simulations, the under- relaxation factors are set as 0.2, 0.5, and 0.9 for pressure, momentum, and energy, respectively. All other under- relaxation factors are retained as their default values.

4.12.1.3. Pressure- Velocity Coupling

Pressure- Velocity Coupling is used to derive an equation for pressure from the discrete continuity equation. There are SIMPLE, SIMPLEC and PISO algorithms

available in FLUENT . The SIMPLE (Semi Implicit Method for Pressure- Linked Equations) algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field. The SIMPLEC (Simple- Consistent) is similar to the SIMPLE algorithm. The only difference lies in the expression used for the face flux correction.

The PISO (Pressure Implicit with Splitting of Operators) scheme is based on the higher degree of approximate relation between the corrections for pressure and velocity. One of the limitations of the SIMPLE and SIMPLEC algorithms is that new velocities and corresponding fluxes do not satisfy the momentum balance after the pressure-correction equation is solved. As a result, the calculation must be repeated until the balance is satisfied. To improve the efficiency of this calculation, the PISO algorithm performs two additional corrections: neighbor correction and skewness correction.

Neighbor Correction: The main idea of the PISO algorithm is to move the repeated calculations required by SIMPLE and SIMPLEC inside the solution stage of the pressure-correction equation. After one or more additional PISO loops, the corrected velocities satisfy the continuity and momentum equations more closely. This iterative process is called a momentum correction or "neighbor correction". The PISO algorithm takes a little more CPU time per solver iteration, but it can dramatically decrease the number of iterations required for convergence, especially for transient problems.

Skewness Correction: For meshes with some degree of skewness, the approximate relationship between the correction of mass flux at the cell face and the difference of the pressure corrections at the adjacent cells is very rough. Since the components of the pressure-correction gradient along the cell faces are not known in advance, an iterative process similar to the PISO neighbor correction described above is desirable. After the initial solution of the pressure-correction equation, the pressure-correction gradient is recalculated and used to update the mass flux corrections. This process, which is referred to as "skewness correction", significantly reduces convergence difficulties associated with highly distorted meshes. The PISO skewness correction allows FLUENT to obtain a solution on a highly skewed mesh

in a approximately the same number of iterations as required for a more orthogonal mesh. In the present investigation, the PISO algorithm with skewness and neighbor corrections is used for all simulations.

4.12.2. Numerical Scheme

In the present investigation, the segregated solution method is used to solve the governing integral equations for the conservation of mass, momentum and energy. In this method, the governing equations on the individual control volumes are integrated for the unknown dependent variables such as velocities, pressures, temperatures, etc. Then the discretized equations are linearized and resultant linear equation system are solved to yield updated variables of the dependent variables.

In the segregated solution method, the governing equations are solved sequentially. Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the steps shown below:

1. Fluid properties are updated based on the current solution.
2. The u , v , and w momentum equations are each solved in turn using current values for pressure and face mass fluxes, in order to update the velocity field.
3. Since the velocities obtained in Step 2 may not satisfy the continuity equation locally, a "Poisson-type" equation for the pressure correction is derived from the continuity equation and the linearized momentum equations. This pressure correction equation is then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass fluxes such that continuity is satisfied.
4. Where appropriate, equations for scalars such as turbulence, energy, species, and radiation are solved using the previously updated values of the other variables.
5. A check for convergence of the equation set is made.

These steps are continued until the convergence criteria are met.

In the segregated solution method, the discrete, non-linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow-field solution. Implicit means: For a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighboring cells. Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities.

In the segregated solution method, each discrete governing equation is linearized implicitly with respect to that equation's dependent variable. This will result in a system of linear equations with one equation for each cell in the domain. Because there is only one equation per cell, this is sometimes called a "scalar" system of equations. A point implicit (Gauss-Seidel) linear equation solver is used in conjunction with an algebraic multigrid (AMG) method to solve the resultant scalar system of equations for the dependent variable in each cell. For example, the x -momentum equation is linearized to produce a system of equations in which u velocity is the unknown. Simultaneous solution of this equation system (using the scalar AMG solver) yields an updated u -velocity field.

In summary, the segregated approach solves for a single variable field (e.g., p) by considering all cells at the same time. It then solves for the next variable field by again considering all cells at the same time, and so on.

4.12.3. Convergence

The convergence criterion for the continuity and momentum (x , y , and z velocities) equations is set as 10^{-3} while it is set as 10^{-6} for the energy residual. All the simulations are performed until the solution is converged with the specified convergence criterions.

4.13. Experimental Study

In the present study, Particle Image Velocimetry (PIV) technique is used to investigate the flow structure in the flow passage of plate fin and tube heat exchangers. PIV can be described as a technique that provides a large number of instantaneous velocity vectors at many points over a plane. Figure 4.16 shows a schematic description of a typical PIV system. All PIV experiments are conducted at the Cukurova University, Mechanical Engineering Department, Fluid Mechanics Research Laboratory.

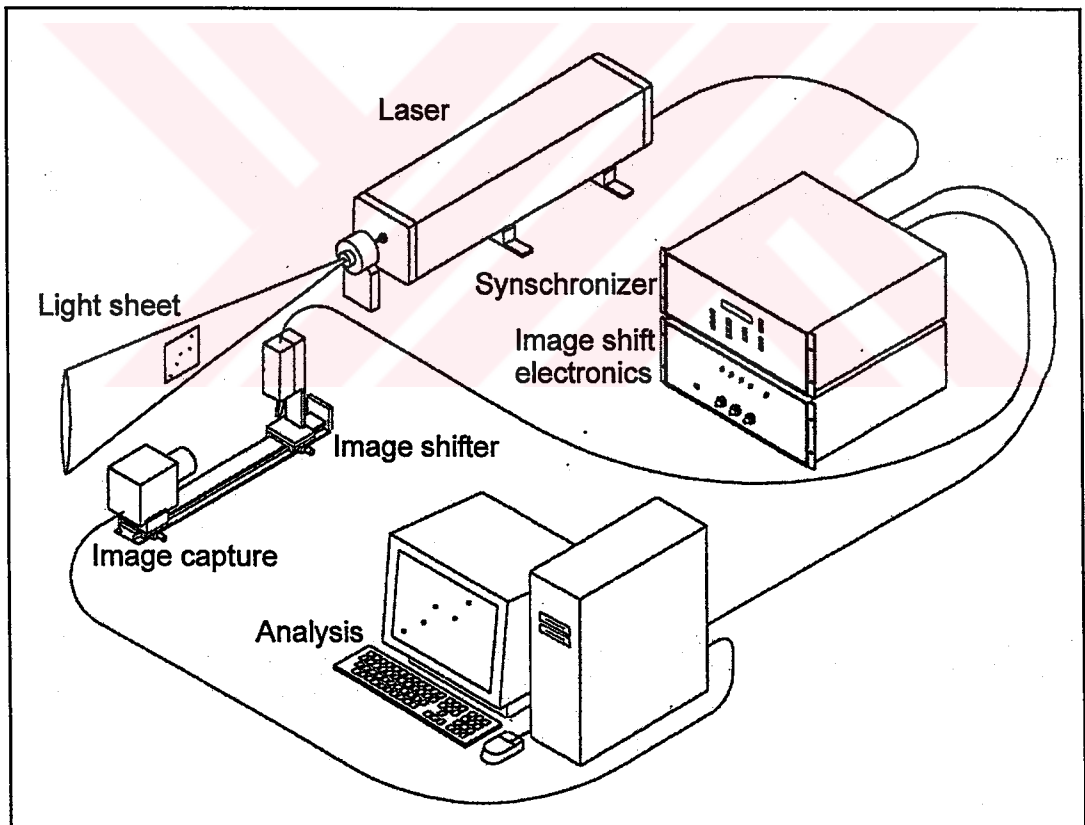


Figure 4.16. Schematic description of a typical PIV system (Akar, 2003)

4.13.1 Experimental Set-up

The test section is placed in a water channel which is seeded with spherical shaped silver coated PVC particles with a mean diameter of 12 μm and has a specific gravity that is the same as of water. The closed-loop open surface water channel is made of clear plexiglass sheet of 15 mm thick and is 8000 mm long, 1000 mm wide, and 750 mm deep. The water channel has a longitudinal free stream turbulence intensity of 0.5%. The desired mean flow in the test section is created by a pump that is driven by an electric motor with varying speed controller.

The flow field is enlightened by using a pair of double-pulsed Nd:YAG laser units. Each laser produces a thin and intense light sheet and one pulse of the pulse pair. By pulsing the light sheet twice, it is possible to acquire a two dimensional vector analysis of the fluid motion by using the change in position of the particles between the two pulses and the time between the pulses. The locations of the particles is recorded and an analysis is performed to determine the velocity vectors of particles throughout the two-dimensional area of the flow. A cross-correlation CCD camera with a resolution of 1024 x 1024 pixels is used to capture the particle images and to transfer the image to a computer program. Crosss correlation is a process which uses two separate images. The video images of the particles are analyzed using powerful processing algorithms to determine the particle displacement that occurs during the time between laser pulses. To transfer the images from the camera to the computer, a high speed digital frame grabber is employed. The frame grabber transfers the data to the computer at the highest possible speed permitted by the PCI bus. The c omputer c alculates t he v ector field b y p rocessing t he l ighted i mages. A software platform "Flow Manager" provides comprehensive solutions for capturing, analyzing, and displaying PIV images. To capture the flow field image, the laser pulse and camera m ust b e t riggered with the correct sequence and timing for the flow conditions under investigation. Therefore, a synchronizer is used to control all of the components are initiated at the exact moment necessary. The power and speed of the processing techniques are such that the velocity field after validation can be obtained on-line. In addition to vorticity and strain rate fields, statistical properties

such as turbulence and mean values can be obtained globally by collecting the desired number of images processing them.

A picture of the experimental set-up can be seen in Fig. 4.17.



Figure. 4.17. A picture of the experimental set-up

4.13.2. Test Section

The test section consists of a tube mechanically press fitted to the two parallel plates. The tube and the plates are made from plexiglass material so that the laser light propagates easily from the plates and the tube. The tube outer diameter and the fin length are 50 mm, and 152.5 mm, respectively. The experiments are conducted for fin spacing of 18.25 mm and 5.8 mm. The corresponding non-dimensional fin spacings are $F_s = 0.365$ and $F_s = 0.116$, non-dimensional fin height is $W = 1.0625$,

and non-dimensional fin length is $L_f = 3.05$. The non-dimensionalization parameter is chosen as the tube diameter. In the test section, the fin height and the fin length are extended 5 times the tube diameter both upstream and downstream of the tube in both longitudinal and transverse directions in order to avoid the flow disturbances that may occur between the plates in the test section. The flow Reynolds number is varying between $600 < Re < 7000$. The depth of the water in the water channel is adjusted to a 65 mm height.

5.15. RESULTS AND DISCUSSION

The design of a heat exchanger involves several considerations with respect to its geometrical and operating parameters. There are a number of companies for designing, marketing and selling heat exchangers throughout the world. Unfortunately, there are no standard parameters concerning geometrical and operating characteristics of the plate fin and tube heat exchangers. The designers even do not want to share their typical exchangers' characteristic geometrical and operating parameters because of their commercial rights. This has motivated us to do an optimization study to find optimum geometrical dimensions of a model heat exchanger.

In the present study, firstly, an experimental verification and a grid independency analysis have been made to demonstrate the validity and the grid independency of the present numerical solution. Secondly, a test case for a single row configuration has been initially performed to understand the local flow and heat transfer characteristics of the plate fin and tube heat exchangers under laminar flow conditions. Following the numerical calculations and their validations against experimental data for a single row configuration, the present numerical model has been extended to simulate multi row configurations. Effects of several geometrical parameters such as fin spacing, number of tube row, tube arrangement, fin configuration, and wavy angle are investigated in detail under laminar flow assumption. Then, a series of simulations concerning transient and turbulent behavior of the flow are performed. Finally, a modified dynamic LES method is applied to the optimum heat exchanger model used in the present work.

5.1. Grid Independency Analysis and Experimental Verification

In the present study, a check for grid independency has been conducted in order to ensure that the simulation results are not grid dependent. To do this check, a heat exchanger model having dimensions of $F_s = 0.233$, $L_f = 19.05$, $W = 12.7$ and comprised of 4 cylinders, staggered layout, wavy fin configuration is selected from

Wang et al. (1997)'s experimental study [sample no: 8]. The geometry and the grid distribution of the test model used in the experimental study of Wang et al. (1997) have been developed using FIDAP Finite Element Code. The grid independency tests were carried out by using different number of grid nodes distributed along the x , y , and z - axes in order to test the effect of grid distribution as well as grid independency in the computational domain.

Figure 5.1 shows f and j factors as a function of Reynolds number obtained for different grid distributions and number of node points. Experimental data of Wang et al. (1997) are also included for demonstrating the validity of the mesh sensitivity analysis as well as experimental verification. The friction factor f and Colburn factor j are known as good design parameters for the compact heat exchangers. Generally, the dimensionless heat transfer characteristics are defined in terms of the j factor as it is directly related to the surface Nusselt number. The friction factor f is equally important because it determines the pressure drop across the heat exchanger. Both factors are calculated as follows:

$$f = \frac{P_{exit} - P_{\infty}}{\rho U_{\infty}^2} \times \frac{F_s}{L_f} \quad (5.1)$$

$$j = \frac{Nu}{Re_D Pr^{1/3}} \quad (5.2)$$

From Fig. 5.1, it is clear that mesh distributions with (188 x 26 x 8) and (228 x 44 x 10) give very similar results and they both overestimate the experimental data in the calculation of the j - factor. It is important here to note that changing the number of node points in the x - and y - directions do not make a big sense on the calculation of the j - factor. Therefore, it is decided to increase the number of grid points in the z - direction. When the number of grid points are increased in the z - direction, (228 x 46 x 20), the calculated results for the j - factor correspond very well with the experimental data as it is clearly seen from the figure. Increasing the node points in the z -direction alters the previous results for the calculated values of the j -

factor and shows that the flow is highly three-dimensional considering other grid distributions.

In order to be sure that this node points are really enough to predict the flow and heat transfer behavior in the heat exchangers, the calculated values of the f -factor are also checked for consistency with the experimental data. Three grid systems containing (228 x 46 x 16), (228 x 46 x 18), (228 x 46 x 20) node points were tested as shown in the figure. For low Reynolds numbers, all of the three grid systems give very close results and correspond well with the experimental data. However, the numerical results slightly underestimate the experimental data when the Reynolds number is greater than 3000. The grid distribution with (228 x 46 x 20) nodes are very consistent with the experimental data and this grid distribution insures us for grid independency of the results.

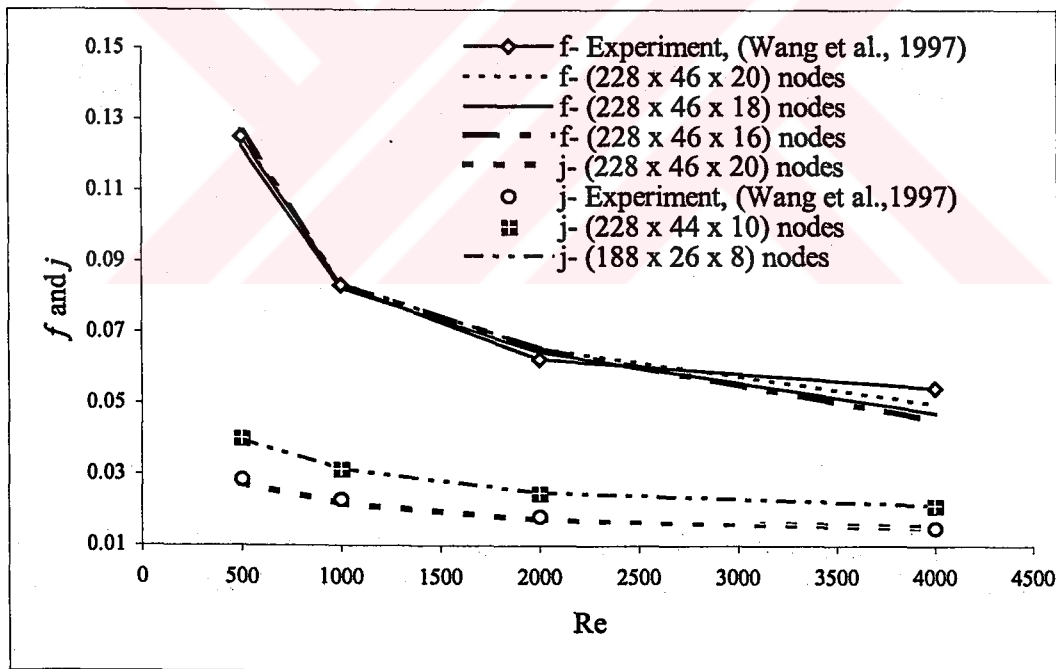


Figure 5.1. Grid independency check

From the mesh sensitivity analysis, it has been observed that an unstructured mesh system containing non-uniformly distributed (228 x 46 x 20) grid points are considered to be fine enough to predict the flow features and corresponding heat transfer characteristics in the heat exchanger passage comprised of four cylinder.

The number of tube rows for the heat exchanger models used in the present study vary from 1 to 5 rows having different geometrical parameters such as fin spacing, tube diameter, fin height, and fin length. In order to provide similar conditions in terms of grid distributions, all heat exchanger models are meshed by using the same constant ratio concerning number of node points per unit of the characteristic geometrical parameter used in the heat exchanger model, i.e. (number of node points / fin spacing), or (number of node points / number of tube row), etc.

The validity of the present numerical study has also been tested against experimental data of Seshimo and Fujii (1991) in terms of heat transfer coefficient as a function of Reynolds number. The geometry of the test model with a plain fin and a single cylinder configuration has a non-dimensional fin length of $L_f = 1.86$, non-dimensional fin height of $W = 2.142$, non-dimensional fin spacing of $F_s = 0.176$, and tube diameter of $D = 9.52$ mm. The inlet frontal velocities are ranging from 0.5 to 3 m/s and the corresponding Reynolds numbers are varying between 900 and 1900. In their study, all physical parameters such as density, specific heat, viscosity and thermal conductivity are taken at a temperature as the arithmetical mean of the inlet and exit air temperatures through the test heat exchanger. Figure 5.2 illustrates heat transfer coefficients obtained from the present numerical simulations and experimental data of Seshimo and Fujii (1991), for the specified heat exchanger model as expressed above. Even if the numerical results are slightly overestimate the experimental data, the results are in good agreement with the experimental data as shown in Fig. 5.2.

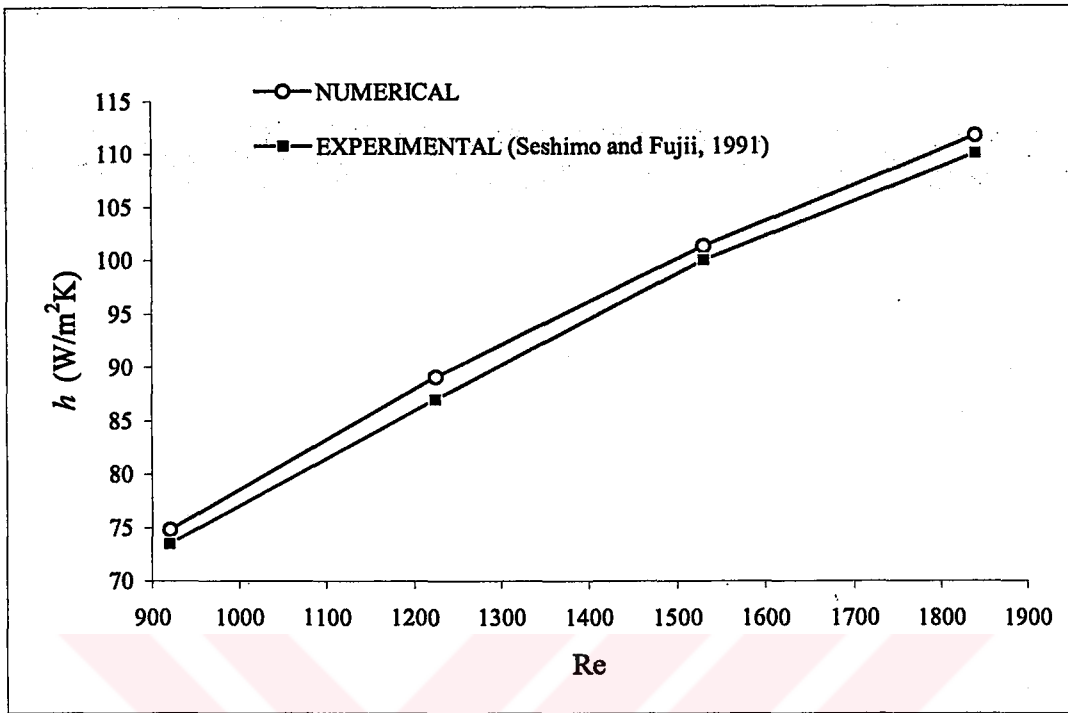


Figure 5.2. Comparison of experimental data and numerical results of heat transfer coefficients as a function of Reynolds number

There are also another comparisons with the experimental studies in terms of horseshoe vortex formation, heat transfer coefficients, flow structures, f and j factors regarding different geometrical parameters and Reynolds numbers in the different sections of the present study.

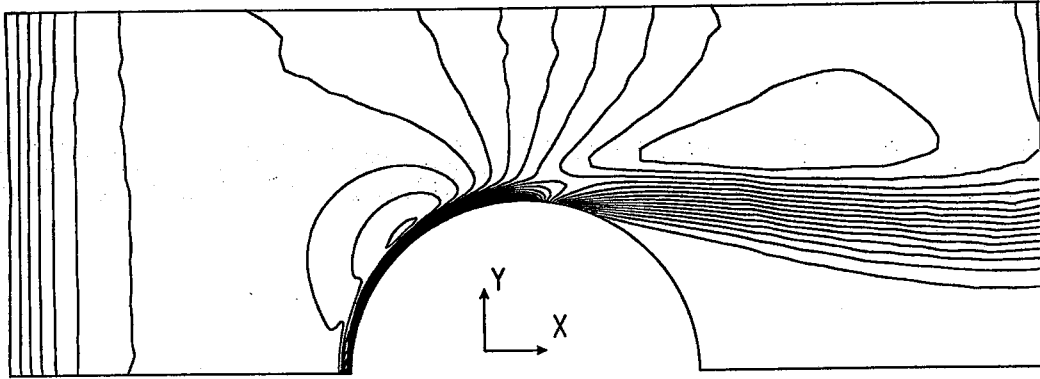
5.2. Results of Optimization Studies for Laminar and Steady State Flow

5.2.1. Effect of Fin Spacing

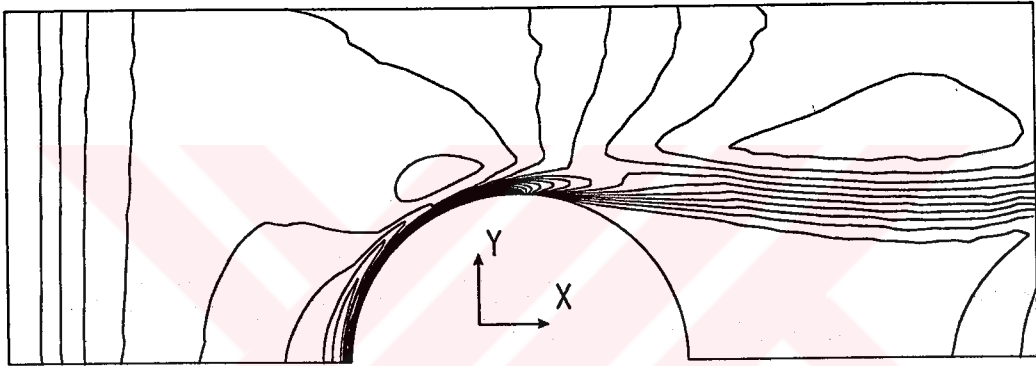
A test case for plain fin and a single cylinder configuration has been performed with a series of steady state simulations for 3-D, incompressible, laminar flow conditions with fin spacings in the $0.03 \leq F_s \leq 0.468$ range for the tube diameter based Reynolds number ranging from 600 to 2000. As it is well- known, the flow structure in a plate fin and tube heat exchanger can be fairly complex and the flow complexities are primarily attributable to the existence of a horseshoe vortex in front

of the tube and a developing boundary layer over the tubes (Mendez et al., 2000; Tsai and Sheu, 1998; Valencia, 1995).

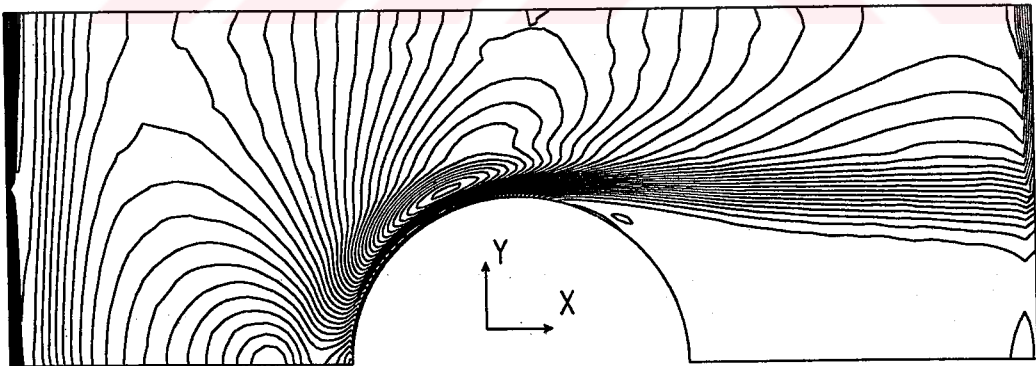
The formation of horseshoe vortex on the forward part of the tube is investigated for varying fin spacing as a function of Reynolds number in comparison with the experimental results of Mendez *et al.*, (2000). Figs. 5.3a-e show the average vorticity contours around the tube surface obtained from the present numerical simulations and the experimental study of Mendez *et al.*, (2000) as an indication of the effect of the fin spacing on the formation of the horseshoe vortex mechanism. The flow is in the x -direction. At a very small fin spacing value of $F_s = 0.116$, numerical simulations neither for $Re = 1200$ nor for $Re = 1460$ are able to reproduce the horseshoe vortex on the forward part of the tube as seen in Figs. 5.3a and 5.3b. As the fin spacing is increased to a value of $F_s = 0.365$ for $Re = 1460$, boundary layers develop from the leading edge of the fin surface and over the tube roll-up into horseshoe vortex as seen in Fig. 5.3c. These numerical results correspond well with those of experimental study of Mendez et al. (2000) at $Re = 1200$ for $F_s = 0.116$ and at $Re = 1460$ for $F_s = 0.365$ as shown in Figs. 5.3d and 5.3e, respectively. It may be concluded from the vorticity contours that the formation of the horseshoe vortex mechanism is dependent more on the fin spacing rather than the Reynolds number for the investigated Reynolds number and fin spacing range.



(a)



(b)



(c)

Figure 5.3. Numerical results for the time averaged vorticity plots for the single cylinder configuration; a) $F_s=0.116$, $Re=1200$, b) $F_s=0.116$, $Re=1460$, c) $F_s=0.365$, $Re=1460$

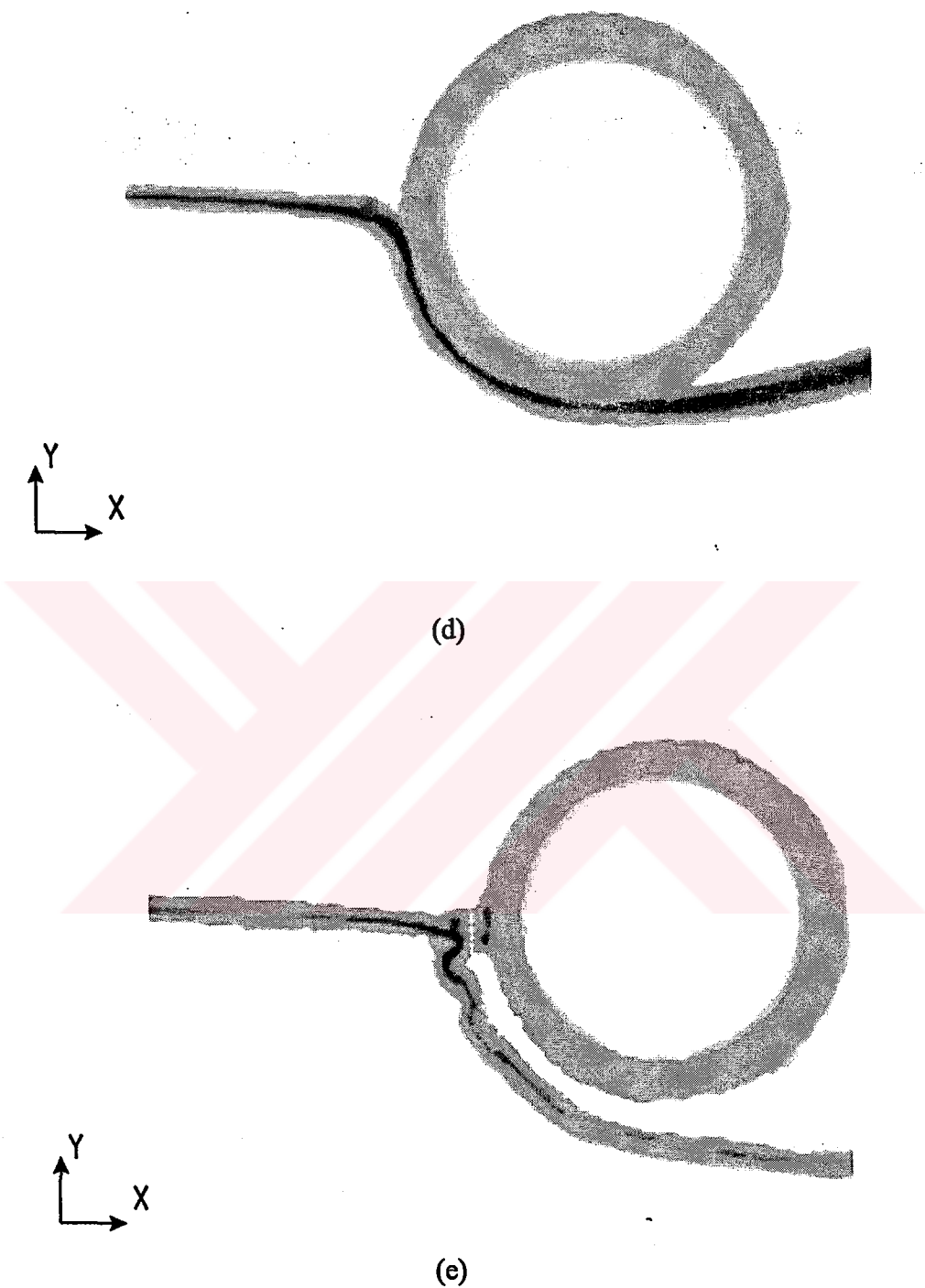


Figure 5.3. (continued) Time averaged vorticity plots for the single cylinder configuration; d) $F_s = 0.116$, $Re=1200$ (experimental data, Mendez *et al.* (2000)), e) $F_s = 0.365$, $Re=1460$ (experimental data, Mendez *et al.* (2000))

The local pressure distributions around the tube surface for a single cylinder configuration, in terms of dimensionless pressure coefficient, C_p , for laminar simulations with varying fin spacing for different Reynolds numbers are shown in Fig. 5.4. The pressure coefficient is defined as follows:

$$C_p = \frac{p_n - p_\infty}{\frac{1}{2} \rho U_\infty^2} \quad \text{.....(5.3)}$$

where p_n is the pressure at $Z = F_s / 2$ points around the tube surface corresponding different θ - angles, p_∞ is the free stream pressure, and U_∞ is the free stream velocity. The angle θ is measured from the forward stagnation point of the tube.

In the present study the local and the average values of the flow and heat transfer data are obtained by different ways. The local data is obtained by creating iso surfaces or point surfaces at the locations in the computational domain where the value of the data is needed. For example, the value of the free stream pressure, p_∞ , in Eq. (5.3) is obtained by creating an iso surface just at the entrance region of the fin and calculating the average static pressure data on this iso surface. Similarly, the local pressure values around the tube surface, which is denoted by p_n in the same equation, are obtained by creating point surfaces at $Z = F_s / 2$ at the corresponding θ angle and calculating the value of the static pressure at the specified point surface.

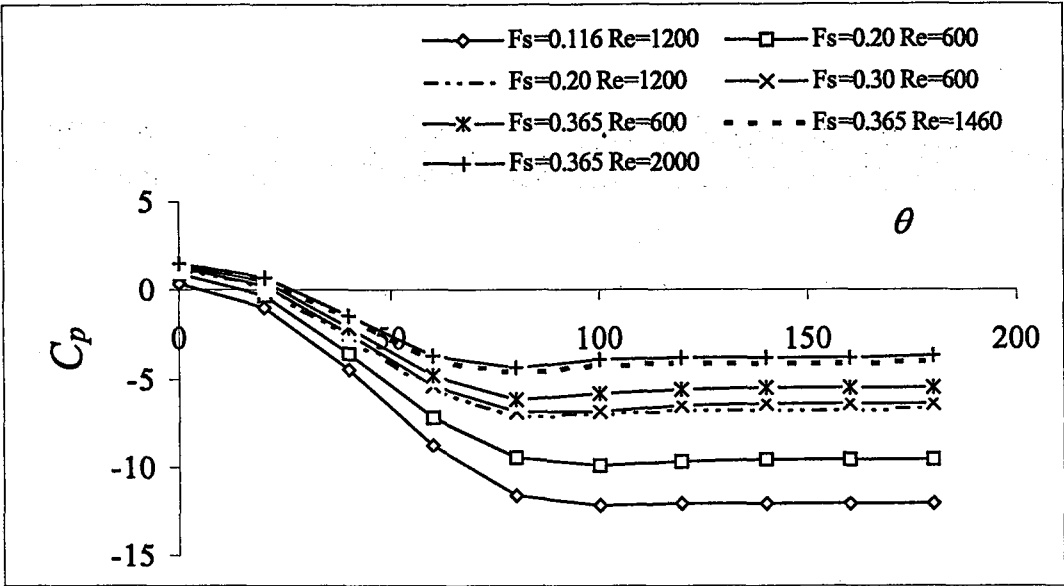
The average data are considered as the area- weighted averaged values of these data and are computed by the present finite volume code (FLUENT, 1998) for both the fin and the tube surfaces from the computation of the surface integrals at the specified surfaces as follows:

$$\frac{1}{A} \int \phi dA = \frac{1}{A} \sum_{i=1}^n \phi_i |A_i| \quad (5.4)$$

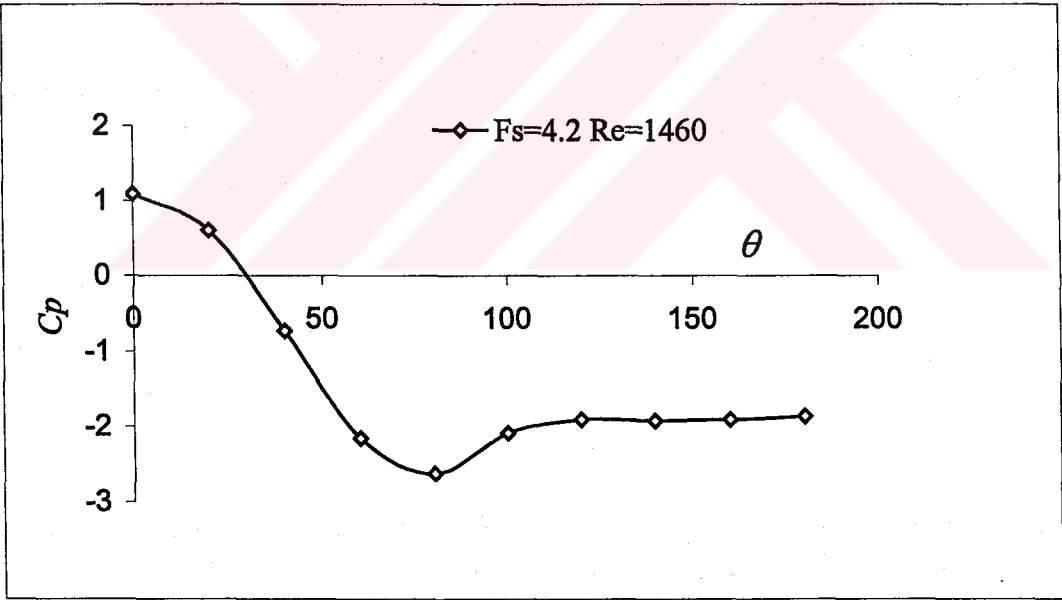
Where ϕ_i is the selected field variable for the flow or heat transfer, A is the total surface area and A_i is the heat transfer surface area.

As it is clearly seen in Fig. 5.4, surface pressure profiles for each numerical simulation look similar in the upstream tube half while there are variations in the downstream half. The calculated local pressure coefficient distributions deviate from each other in the downstream half depending upon the variations in the fin spacing and give different base pressure coefficient values. As the fin spacing is increased from $F_s = 0.20$ to $F_s = 0.25$ and to $F_s = 0.30$ at $Re = 600$, the calculated base pressure coefficient values (which is the value of the C_p at the angle of $\theta = 180^\circ$) also increase due to the pressure difference increase in the wake of the tube. These calculated base pressure coefficient values for $F_s = 0.20$, $F_s = 0.25$, $F_s = 0.30$ and $F_s = 0.365$ are found to be -7.07 , -6.13 , -5.54 , and -5.06 , respectively for $Re = 600$. When the Reynolds number is further increased to a value of $Re = 1200$ for the fin spacing values of $F_s = 0.116$ and $F_s = 0.20$, an increase from a value of -8.15 to -5.64 for the base pressure coefficient is observed. When the Reynolds number is further increased to a value of $Re = 1460$ and $Re = 2000$, the base pressure coefficient values become larger compared to those corresponding values obtained at $Re = 600$ and $Re = 1200$. An increase from a value of -4.31 to -3.52 for the base pressure coefficient is observed when the Reynolds number is increased from $Re = 1460$ to $Re = 2000$ for $F_s = 0.365$. As the fin spacing is increased, the width of the wake region also increases and the flow separation in the wake with a recirculation zone occurs further downstream of the tube. This recirculation zone becomes larger allowing reentrant of the core flow through the passage. As a result, higher base pressure coefficient values are obtained. The pressure difference between the inlet and outlet sections of the heat exchanger again decreases when the fin spacing is increased in the same range of $0.116 \leq F_s \leq 0.365$. The overall comparison between these eight simulations show that the calculated base pressure coefficient values increase as the Reynolds number and fin spacing are both increased but the effect of fin spacing seems to be more influential on the sensitivity of the local flow than the Reynolds number for the investigated Reynolds number and the fin spacing range ($600 \leq Re \leq 2000$ and $0.116 \leq F_s \leq 0.365$).

When the fin spacing is further increased to a value of $F_s = 4.2$ at $Re = 1460$, the local pressure distribution along the cylinder surface at $Z = F_s / 2$ is found to be that around a single cylinder, as shown in Fig. 5.4b. In this case, the fin spacing is much larger than the tube diameter, and the hydrodynamics is not affected by the presence of the fin, because and the length of the tube is too high. That's why the values of the pressure coefficients become as a flow around a single cylinder alone. This fact explains why the calculated values for the pressure coefficients around the cylinder in the heat exchanger flow passage are found very small compared with that of a single cylinder.



(a)



(b)

Figure 5.4. (a) Local pressure coefficients around the tube surface for the single cylinder configuration with varying fin spacing and Reynolds number conditions
(b) Local pressure coefficients around the tube surface for $F_s = 4.2$, and $Re = 1460$

Figure 5.5 shows numerical results of variation of average heat transfer coefficient as a function of fin spacing for $Re = 920$. The average heat transfer coefficient is obtained as the computation of the surface integrals on the fin and the tube surfaces, as shown in Eq. (5.4), since the tube and the fin surfaces both correspond to the heat transfer surface area. As indicated in the figure, it is clearly seen that as the fin spacing is increased there is a sharp increase in the average heat transfer coefficient for $F_s \leq 0.15$. In the range $0.15 \leq F_s \leq 0.25$, the average heat transfer coefficient again increases gradually. For $F_s \geq 0.35$, the average heat transfer coefficient is nearly unchanged.

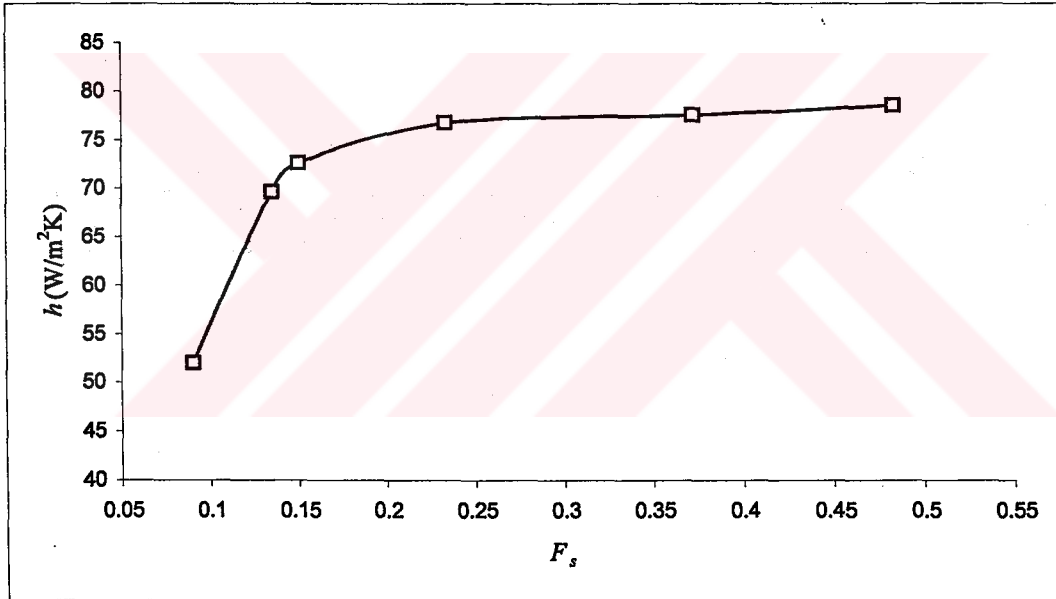


Figure 5.5. Average heat transfer coefficient as a function of fin spacing for $Re=920$

Figure 5.6 shows coefficient of pressure drop between the inlet and outlet sections of the heat exchanger as a function of fin spacing for $Re = 920$. The coefficient of pressure drop is calculated as

$$K = \frac{P_{\infty} - P_{exit}}{\frac{1}{2} \rho U_{\infty}^2} \quad (5.5)$$

where p_{in} and p_{exit} are the pressure at the inlet and exit sections of the heat exchanger, respectively. These pressures are obtained by firstly creating iso surfaces at the inlet and exit sections of the heat exchanger, secondly, getting average static pressures at the specified surfaces. From the figure, it is possible to see three different pressure drop regions when different fin spacing ranges are taken into account. It is clear that as the fin spacing is increased, the pressure drop decreases sharply for $F_s \leq 0.15$ and $Re = 920$. Between $0.15 \leq F_s \leq 0.25$ the pressure drop again decreases gradually. Finally, for $F_s \geq 0.35$ the pressure drop seems to get a constant value for constant Re .

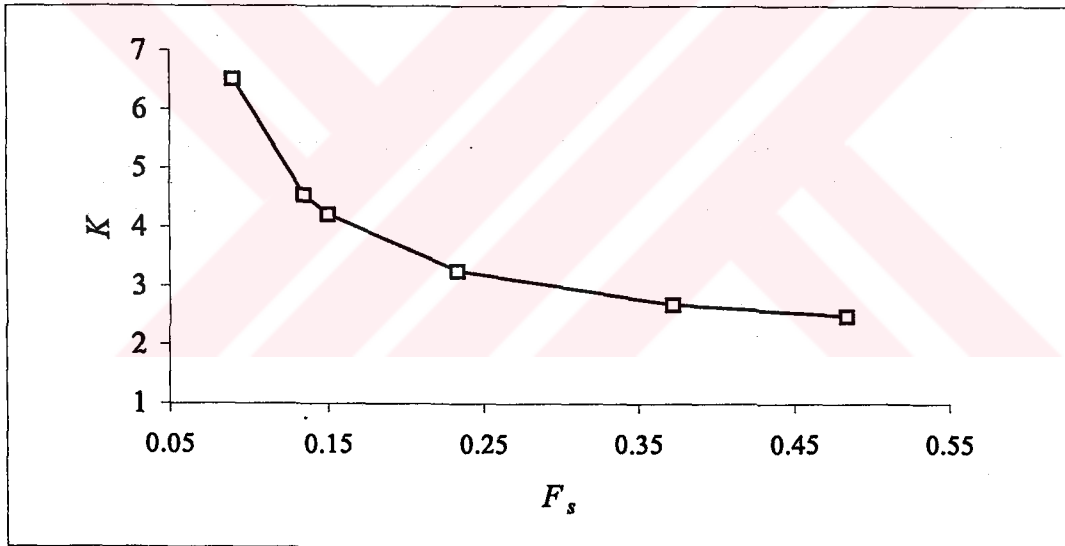


Figure 5.6. Pressure drop coefficient between the inlet and outlet sections of the heat exchanger as a function of fin spacing for $Re = 920$

Mendez et al. (2000) stated that a good design criterion may be based on;

$$\overline{Nu}/(KF_s) \quad (5.5)$$

that is, the heat transfer per unit length of the tube and per unit pressure drop, since this will take into account both compactness and pumping requirements for the heat

exchanger. In Eq. (5.5) $\overline{Nu}$ is the overall Nusselt number and the authors define it as follows:

$$\overline{Nu} = \text{Re Pr} \frac{WF_s}{A_f / d^2 + \pi F_s} \overline{T}_{exit} \quad (5.6)$$

where A_f / d^2 is the nondimensional fin surface, πF_s is the nondimensional tube surface, WF_s is the nondimensional flow area and $\overline{T}_{exit}$ is the average temperature at the exit section.

Figure 5.7 illustrates $\overline{Nu} / (K.F_s)$ as a function of fin spacing, F_s , for $\text{Re} = 600$ and $\text{Re} = 2000$. It is clearly seen from Fig. 5.7 that as the fin spacing is increased, $\overline{Nu} / (K.F_s)$ increases and much higher $\overline{Nu} / (K.F_s)$ values are obtained for $\text{Re} = 2000$ compared to those for $\text{Re} = 600$. This is due to the fact that heat transfer rate increases and pressure drop decreases with the increase in the Reynolds number and the most significant effect is due to the developing boundary layers in front of the tube which lead to the larger mixing flow conditions at higher Reynolds number. For $\text{Re} = 600$ and $\text{Re} = 2000$, the value of $\overline{Nu} / (K.F_s)$ increases sharply with F_s in the $0.116 \leq F_s \leq 0.20$ range where the portion of the tube in the flow passage is very small and the fin surface has a primary importance in the calculation of $\overline{Nu} / (K.F_s)$. In the $0.20 \leq F_s \leq 0.25$ range, there is a slower increase in the $\overline{Nu} / (K.F_s)$ for $\text{Re} = 600$ and $\text{Re} = 2000$. But after a certain value of fin spacing, ($F_s > 0.25$ for $\text{Re} = 2000$), $\overline{Nu} / (K.F_s)$ values start to decrease with F_s and the combined effect of tube and fin surfaces on the calculated value of $\overline{Nu} / (K.F_s)$ is observed. It is important to indicate here that for small fin spacing values, $\overline{Nu} / (K.F_s)$ increases while for larger fin spacing values, it decreases. Thus there is an optimum fin spacing range corresponding to the maximum $\overline{Nu} / (K.F_s)$. This optimum range cannot be seen

clearly for $Re = 600$. Then the optimum value of F_s for the present study can be considered to be in the $0.20 \leq F_s \leq 0.30$ range for the investigated Reynolds numbers.

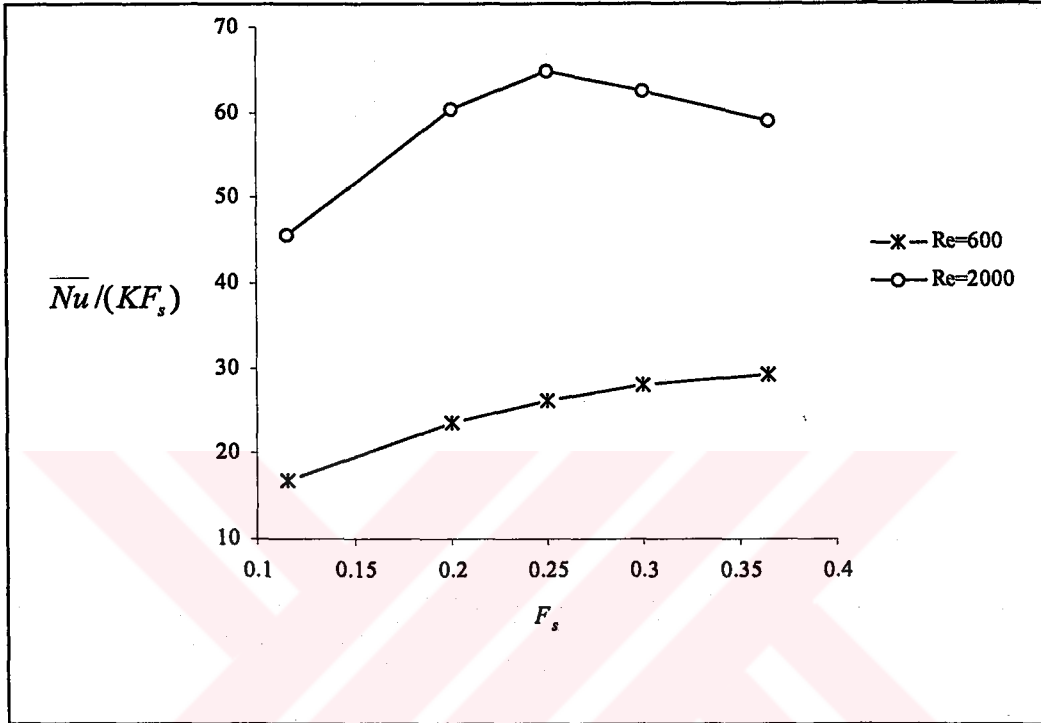


Figure 5.7. $\overline{Nu}/(KF_s)$ as function of F_s for $Re = 600$ and $Re = 2000$ for the single cylinder configuration

Therefore in the present study it is decided to select an optimum fin spacing value regarding both heat transfer and pressure drop data. The optimum fin spacing should correspond to minimum pressure drop and maximum heat transfer data. Considering Figs. 5.5 and 5.6, the optimum fin spacing range may be selected as $F_s \geq 0.20$ where the heat transfer coefficient reaches its highest value and the pressure drop decreases in this range of fin spacing. However, these two figures are not sufficient to optimize the geometry of heat exchangers. But Fig. 5.7 presents the optimum value for the fin spacing in the present study and it is selected as $F_s = 0.233$ for the investigated Reynolds numbers and fin spacing values. This selected fin spacing value also corresponds to the commercial heat exchangers used in the industry which is $0.128 \leq F_s \leq 0.240$ as stated by Mendez et al., (2000).

5.2.2. Effect of Number of Tube Rows

The effect of tube row numbers on the heat transfer performance of the heat exchanger for the selected optimum fin spacing value of $F_s = 0.233$ is investigated for $Re = 1500$ and $Re = 5000$ (Fig. 5.8). The test models used in the simulations with varying tube row numbers and having inlined tube arrangement are all shown in Fig. 4.6- to- 4.10. It is clearly seen from Fig. 5.8 that the heat transfer coefficient increases with the increase in the Reynolds number and decreases as the tube row number is increased. Higher heat transfer coefficient are obtained for the smaller number of tube rows due to the fact that when the number of tube row is low, a longer boundary layer developing from the beginning of the fin surface may result in better heat transfer performance. In addition, a large amount of ineffective area that does not take part in the transfer of heat may occur on the fin surface in the case of higher tube row numbers. One can also easily see from Fig. 5.8 that the effect of tube row number on the heat transfer coefficient becomes comparatively small as the tube row number is greater than 4. Moreover, the fact that heat exchanger pressure drop is highly increased by increasing the number of tube rows, as shown in Fig. 5.9, should also be taken into consideration. For this reasons, 4 cylinder configuration could be regarded as an optimum multi-row fin and tube heat exchanger and the rest of the numerical calculations in the present study have been carried out for 4 cylinder configuration.

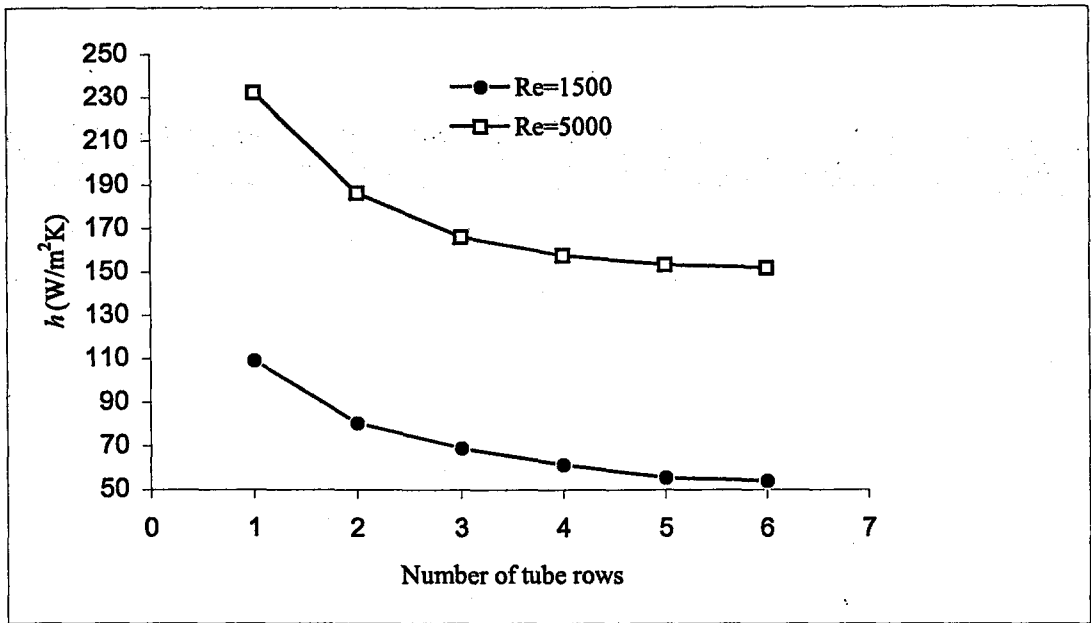


Figure 5.8. Average heat transfer coefficient on the fin surface as a function of number of tube rows for $F_s = 0.233$, $Re=1500$ and $Re=5000$, inlined arrangement

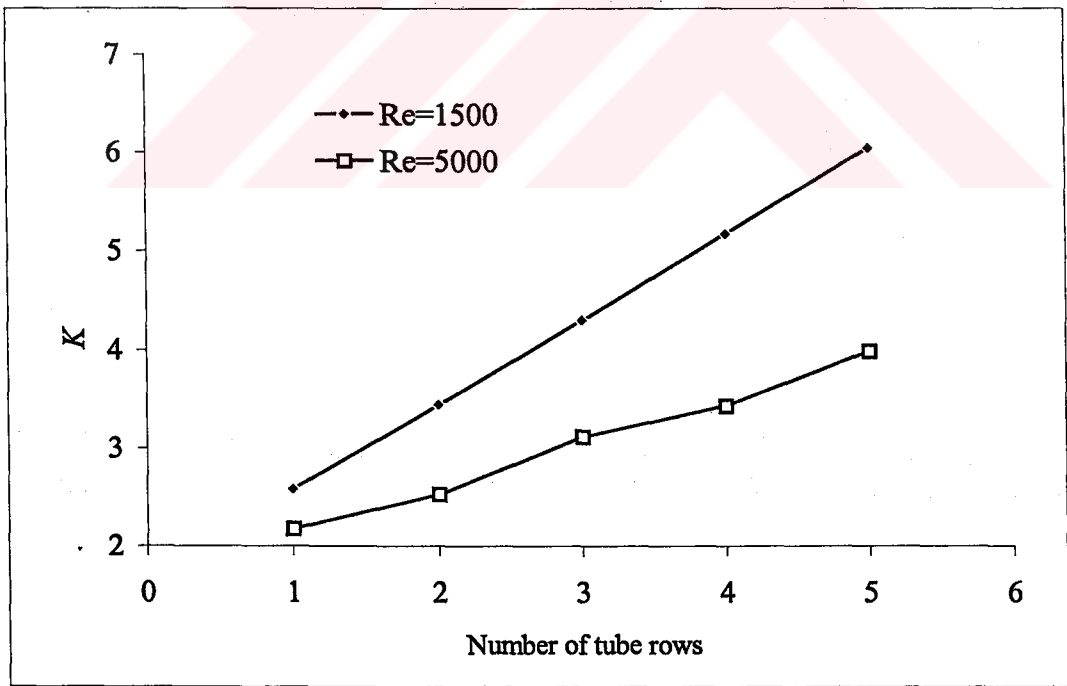
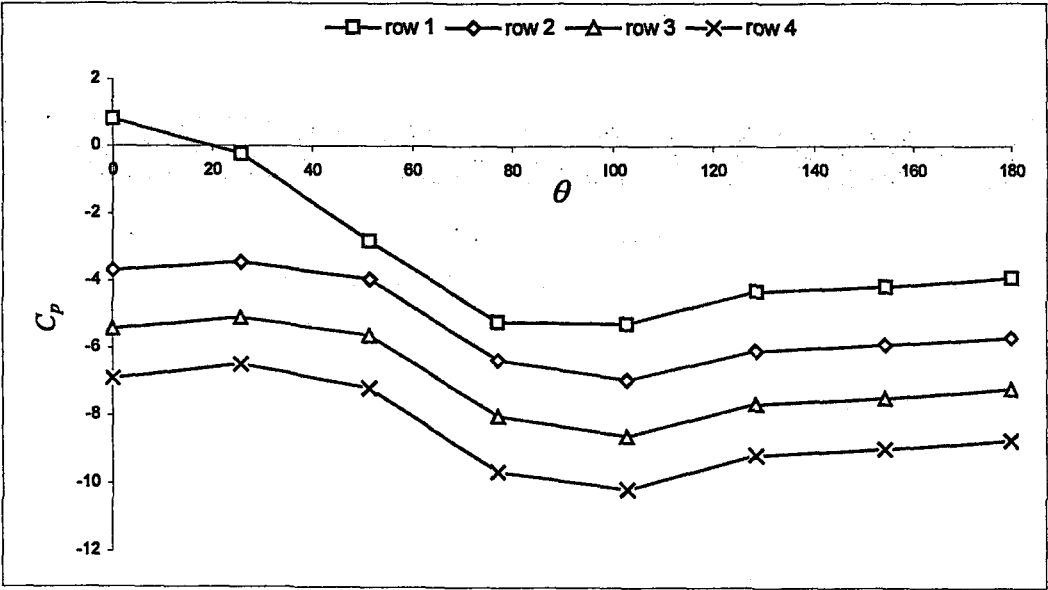


Figure 5.9. Pressure drop between inlet and outlet sections of the heat exchanger as a function of number of tube rows $F_s = 0.233$, $Re=1500$ and $Re=5000$, inlined arrangement

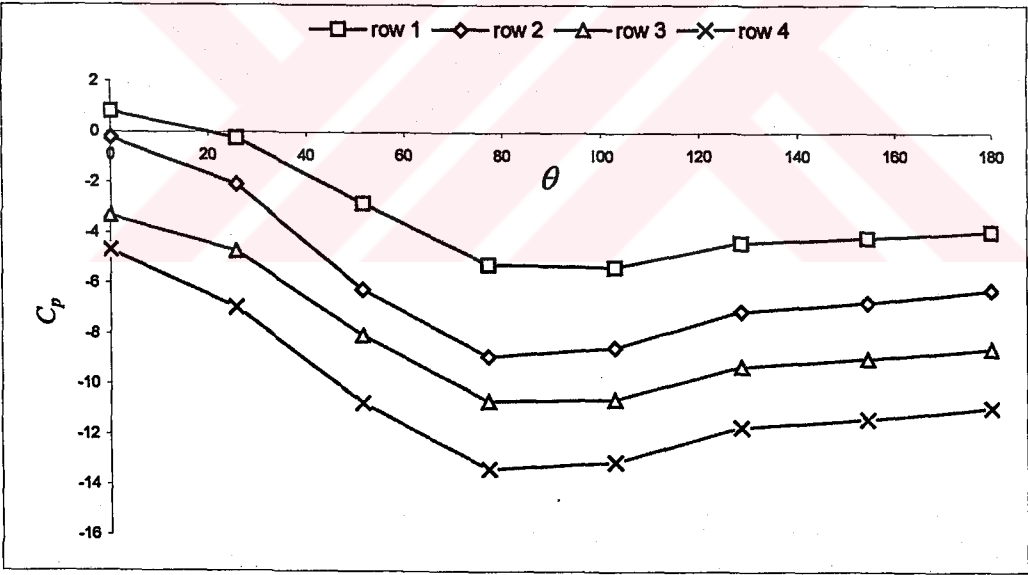
5.2.3. Effect of Cylinder Layout (Inlined- Staggered)

In the present day heat exchanger machines, generally inlined and staggered cylinder layouts are used. Both layouts have different features in terms of flow dynamics in the heat exchanger passage. The inlined layouts are preferred because they prevent dust and fouling in the heat exchanger flow passages. On the other hand, the staggered layouts are able to increase the turbulence and the heat transfer coefficient, and can serve more compact structure compared to the inlined layouts.

In the staggered and inlined cylinder layouts, flow around a tube in the first cylinder is similar to the flow around a single cylinder but the flow pattern in subsequent cylinders is different (Zukauskas, 1974). The hydrodynamics and heat transfer characteristics are greatly affected by the flow patterns that occur in the subsequent cylinders. Therefore numerical simulations for both inlined and staggered layouts (Figs. 4.10 and 4.11) having 4 cylinders for the optimum fin spacing value of $F_s = 0.233$ and for the fin spacing based Reynolds number $Re_s = 600$ have been carried out in order to demonstrate the effect of tube layout on the heat transfer and pressure drop characteristics. Figures 5.10a and 5.10b show the local pressure coefficient distributions around the tube surfaces for inlined and staggered layouts, respectively, at $Z = 0.30$. It is clearly seen that variations of surface pressure profile for each cylinder look similar and the pressure drop increases as the cylinder number is increased for both inlined and staggered layouts. It is found that the magnitude of the pressure drop in the staggered layout is higher than that in the inlined layout. The base pressure coefficient value for the inlined and the staggered layouts is calculated as -8.69 and -11.02 , respectively, for the fourth cylinder.



(a)



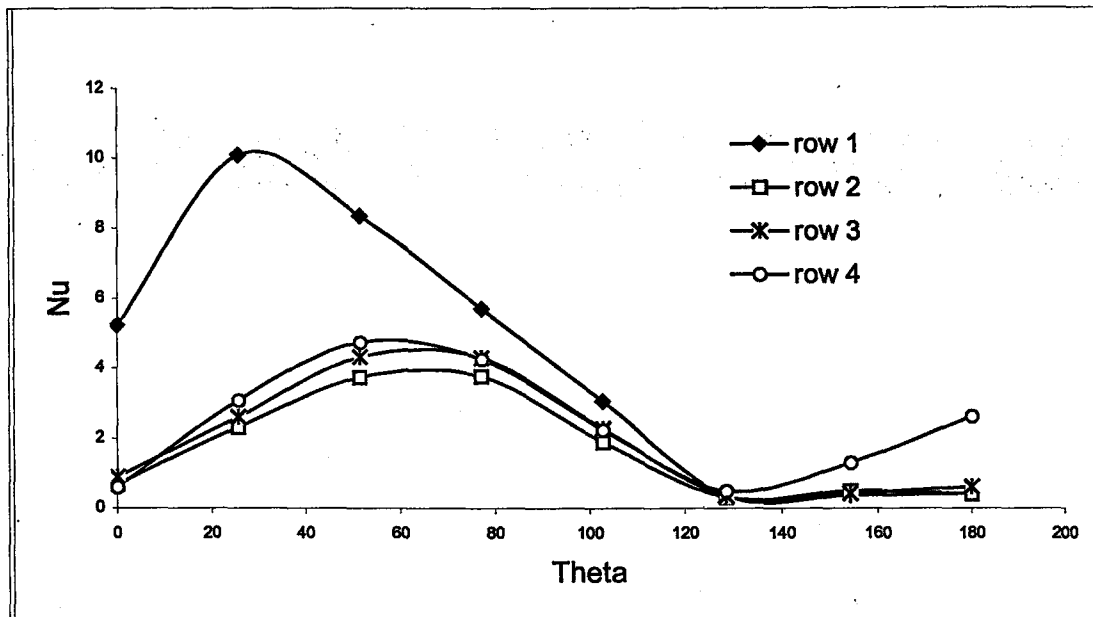
(b)

Figure 5.10. The local pressure distributions around the tube surfaces for the multi row configuration at $Z = 0.20$ for $F_s = 0.233$ and $Re_s = 600$. a) Inlined layout; b) Staggered layout

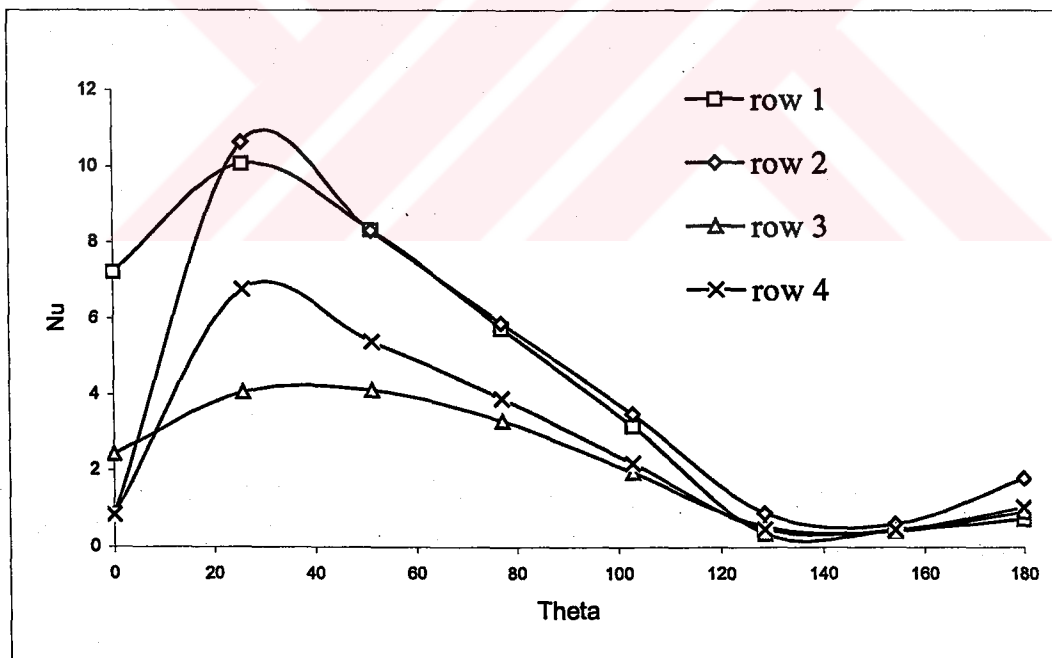
The variations of the local Nusselt number, Nu as a function of the angle θ around the tube surface from the 1st to the 4th cylinder at $Z = 0.20$ for both inlined and staggered layouts are shown in Fig. 5.11a and 5.11b, respectively. The numerical calculations are carried out for $F_s = 0.233$ and $Re_s = 600$. The local heat transfer coefficient, h can be expressed in the non-dimensional form by the local Nusselt number defined as follows:

$$Nu = \frac{hs}{k} \quad (5.7)$$

It is seen in Fig. 5.11a that, for the inlined layout, the local Nusselt number has a maximum value at $\theta = 26^\circ$ for the 1st cylinder, while for cylinders 2 to 4 the maximum Nu occurs at about $\theta = 52^\circ$. On the other hand, for the staggered layout, the maximum Nu occurs at 30° for the 2nd cylinder and its location corresponds to that for the 1st cylinder as seen in Fig. 5.11b. It is also found that the peak value obtained for the staggered layout is higher than that for the inlined layout. Based on Figs. 5.11a and b, it can be said that the maximum value of Nu may correspond to the location of horseshoe vortex system occurs in front of the tube. In the wake of the tube, the local Nusselt number becomes much smaller due to dead flow zone caused by the recirculation region, which indicates poor heat transfer in that region.



(a)



(b)

Figure 5.11. The variations of the local Nusselt number, Nu around the tube surface at $Z=0.20$, for $F_s=0.233$ and $Re_s=600$ for the 4 cylinder configuration; a) Inlined layout; b) Staggered layout

The results of the average heat transfer coefficient, h , calculations as a function of Reynolds number for each layout are presented in Fig. 5.12. The experimental data of Jang et al. (1996) are also included to verify the validities of the present 3-D laminar modelling and the numerical predictions. The average values of the heat transfer coefficients are computed for both the fin and the tube surfaces from the computation of the surface integrals at the specified surfaces by using Eq.(5.4). Fig. 5.12 shows that the larger the Reynolds number, the greater is the heat transfer coefficient for both layouts. The average heat transfer coefficient data for the staggered layout are found to be higher than those for the inlined layout. This is mainly due the fact that the staggered layout provides a better mixing of the fluid in the flow passage compared to the inlined layout. A better mixing of the hot and the cold fluids in the flow passage cause increased heat transfer coefficients as shown in Fig. 5.12. At the same time, it is also shown once more that the present numerical modelling can be considered to be fine enough to predict the heat transfer and pressure drop characteristics for the multi row plate fin and tube heat exchanger.

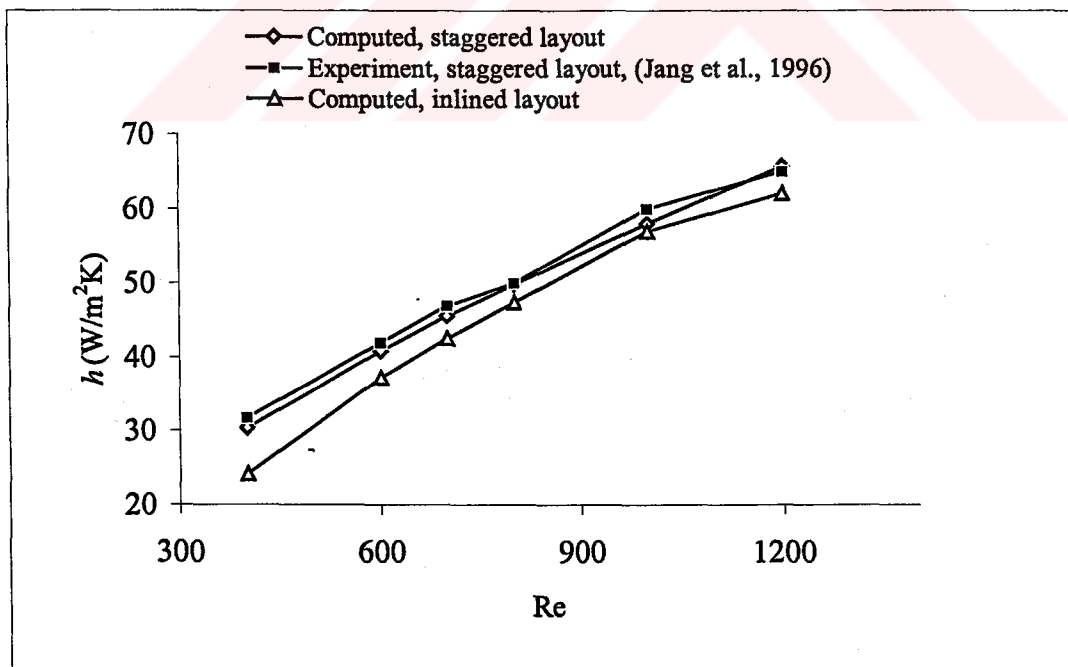


Figure 5.12. Average heat transfer coefficient as a function of Reynolds number for staggered and inlined layouts for the multi row configuration (Misag-147, 2003)

5.2.4. Effect of Fin Configuration (Plain, Wavy)

In order to obtain optimum heat exchanger geometry, the present study has been further extended to investigate effect of fin configuration. In the plate fin and tube heat exchangers various fin configurations such as wavy, louver, convex louver, slit, etc. are used in order to get higher heat transfer performance. Among them, wavy fin surface can lengthen the air flow inside the heat exchanger passage and cause better air flow mixing and thus increased heat transfer performance is expected. In order to investigate the effect of fin configuration on the heat transfer performance, simulation results obtained for four cylinders, staggered/ inlined layouts, wavy and plain fin configurations are compared with each other (Table 5.1). From the Table 5.1 it is clearly seen that higher heat transfer performance is obtained for staggered tube layout and wavy fin configuration.

Table 5.1. Comparison of heat transfer coefficients obtained for four cylinder, plain, wavy, inlined and staggered layouts

Cylinder Layout and Fin Configuration	$h(w/m^2K)$
Inlined layout - Plain fin configuration	61.44
Staggered layout- Plain fin configuration	80.72
Staggered layout- Wavy fin configuration	81.80

5.2.5. Effect of Wavy Angle

The performance of the wavy fin configurations are strongly related to their wavy angles. Wavy angle is sometimes referred as waffle height in the literature. A series of test simulations have been carried for wavy angle ranging from 10° to 30° . In the present investigation affects of different wavy angles are also examined and it is decided to use a wavy angle of 16° for the present investigation.

5.3. Effects of Reynolds Number and Laminar/Turbulence Modelling on the Prediction of Flow Characteristics

As it is outlined in the background section of this study, there is no clear understanding on whether the flow in the heat exchanger passage is laminar or low Reynolds number turbulent flow. Therefore, in the present investigation, both laminar and turbulence models are employed in order to understand the flow and heat transfer characteristics at varying Reynolds number values. Figure 5.13a-e shows velocity vector, streamline, and heat transfer data at $Z = F_s/12$ plane (very close to the fin surface), around the first and the second cylinders, for the four cylinder and wavy fin configuration, staggered layout, having optimum dimensions selected in the previous sections. The Reynolds number ranges from 100 to 1500 ($100 < Re < 1500$). It is clearly seen that heat transfer coefficients increase when the Reynolds number is increased. Higher heat transfer coefficients are obtained in front of the cylinders due to the existence of the horseshoe vortex and lower heat transfer coefficients are obtained in the wake of the cylinders due to the dead flow zone. If the two simulations at $Re = 1500$ for laminar model and RNG $k-\epsilon$ model results are compared with each other, higher heat transfer coefficients in front of the first and the second cylinders are obtained from laminar simulations, as shown in Fig. 5.13d and Fig. 5.13e. In front of the first and the second cylinders, the laminar model gives heat transfer coefficients of $100.672 \text{ W/m}^2\text{K}$ and $125.8396 \text{ W/m}^2\text{K}$, respectively, while RNG $k-\epsilon$ model gives $88.0694 \text{ W/m}^2\text{K}$ and $66.0694 \text{ W/m}^2\text{K}$, respectively. A focus of separation with its saddle point behind each cylinder is observed when the Reynolds number is greater than 500 (Fig. 5.13b-e).

Figure 5.14a-e shows velocity vector and corresponding streamline patterns around the third and the fourth cylinders at $Z = F_s/12$ plane for the optimum heat exchanger for $100 \leq Re \leq 1500$. At $Re = 100$, the horseshoe vortex formation in front of the cylinders is not observed and only a very small recirculation region behind the fourth cylinders is observed (Fig. 5.14a). The length of the recirculation region increases as the Reynolds number is increased as it is clearly seen in Fig. 5.14a-d.

For $Re \geq 500$, a focus of separation with its saddle point behind the third cylinder and a recirculation zone behind the fourth cylinder can be clearly seen (Fig. 5.14b-e). The formation of these flow patterns depend strongly on the value of the Reynolds number as they are not observed at $Re = 100$. Different flow patterns are also observed when the laminar model and RNG $k-\varepsilon$ model simulations at $Re = 1500$ are compared with each other (Fig.5.14d-e). Much bigger areas of the horseshoe vortices are observed in front of the third and the fourth cylinders for the laminar model compared to the RNG $k-\varepsilon$ model. In addition, much larger length of the recirculation region behind the fourth cylinder is obtained for the laminar model compared to the RNG $k-\varepsilon$ model at $Re = 1500$.

Figure 5.15a-e shows velocity vector and streamline pattern at $Z = F_s / 2$ plane (in the middle of the heat exchanger flow passage) around the third and the fourth cylinders between $100 \leq Re \leq 1500$ for the optimum heat exchanger model. The flow patterns are similar to the ones obtained for $Z = F_s / 12$ plane in that as the Reynolds number is increased, the length of the recirculation region also increases. If the laminar model and the RNG $k-\varepsilon$ model simulations at $Re = 1500$ are compared with each other (Fig. 5.15d-e), much smaller recirculation length is observed behind the fourth cylinder for the RNG $k-\varepsilon$ model simulation.

If the flow patterns obtained at $Z = F_s / 12$ and $Z = F_s / 2$ planes are compared with each other, (Fig. 5.14 and 5.15), it can be concluded that, there is no significant difference between the two planes at $Re = 100$ in terms of velocity vector and streamlines. But it is interesting to note that for the laminar model simulations at $Re = 500, 100$, and 1500 , much larger area for the recirculation region behind the third and the fourth cylinders are obtained at $Z = F_s / 2$ plane. Moreover, the horseshoe vortex area in front of the cylinders are very clearly seen at $Z = F_s / 12$ planes, while they cannot be seen at $Z = F_s / 2$ plane.

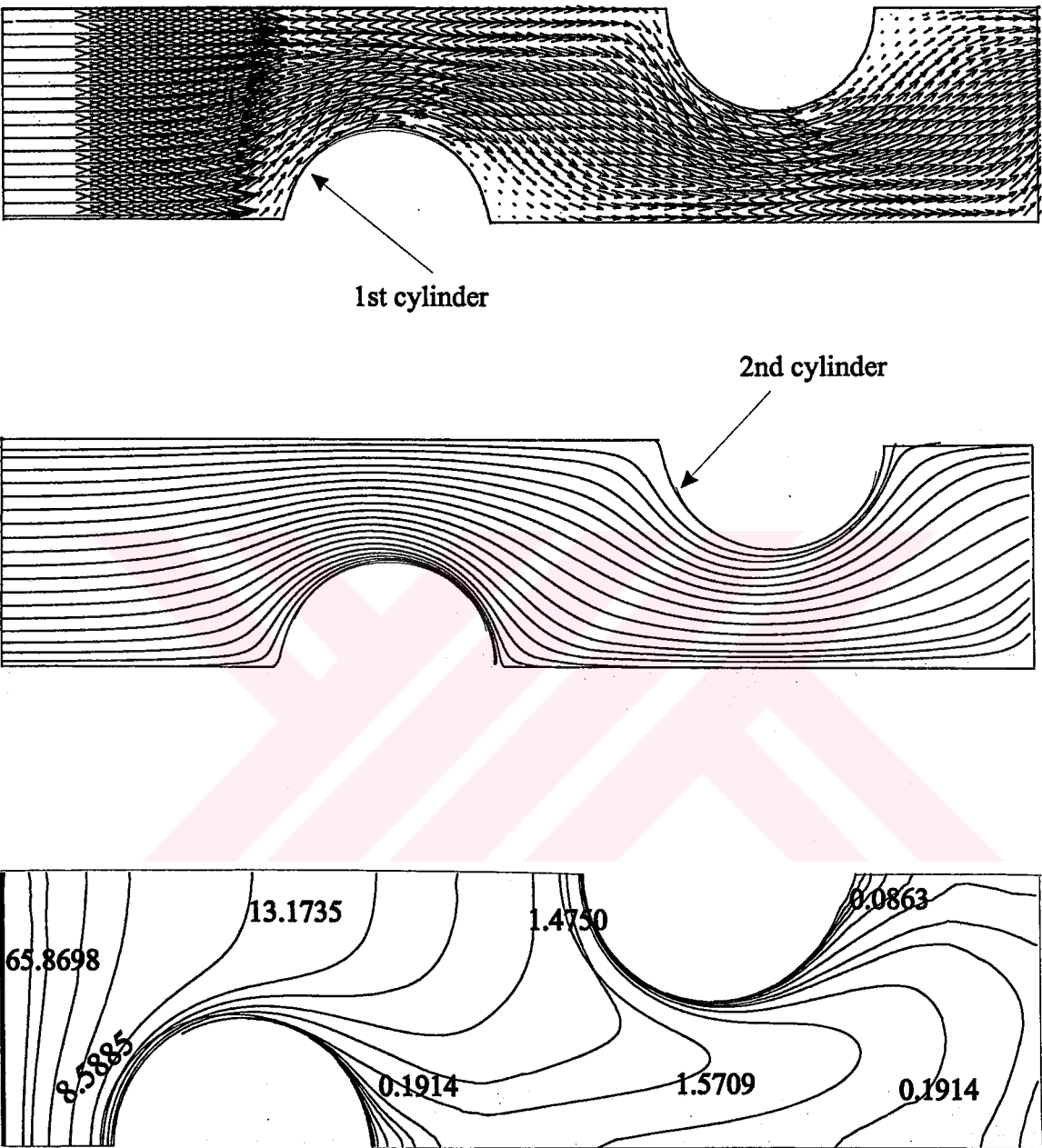


Figure 5.13a. Velocity vector, streamline and heat transfer data at $Z=Fs/12$ plane (very close to the fin surface) around the first and the second cylinders for the wavy fin configuration at $Re = 100$, laminar model results

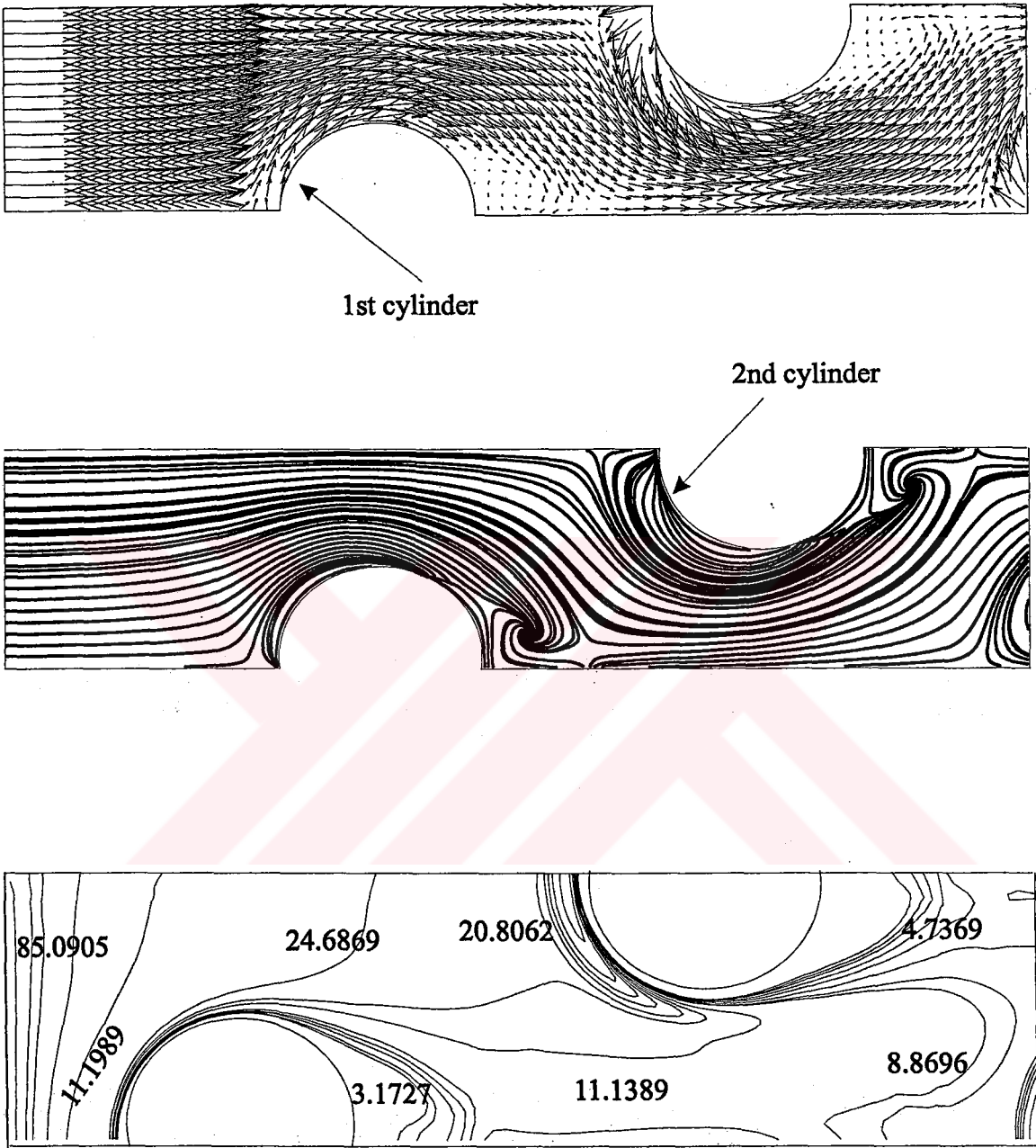


Figure 5.13b. Velocity vector, streamline and heat transfer data at $Z=Fs/12$ plane (Very closeto the fin surface) around the first and the second cylinders for the wavy fin configuration at $Re = 500$, laminar model results

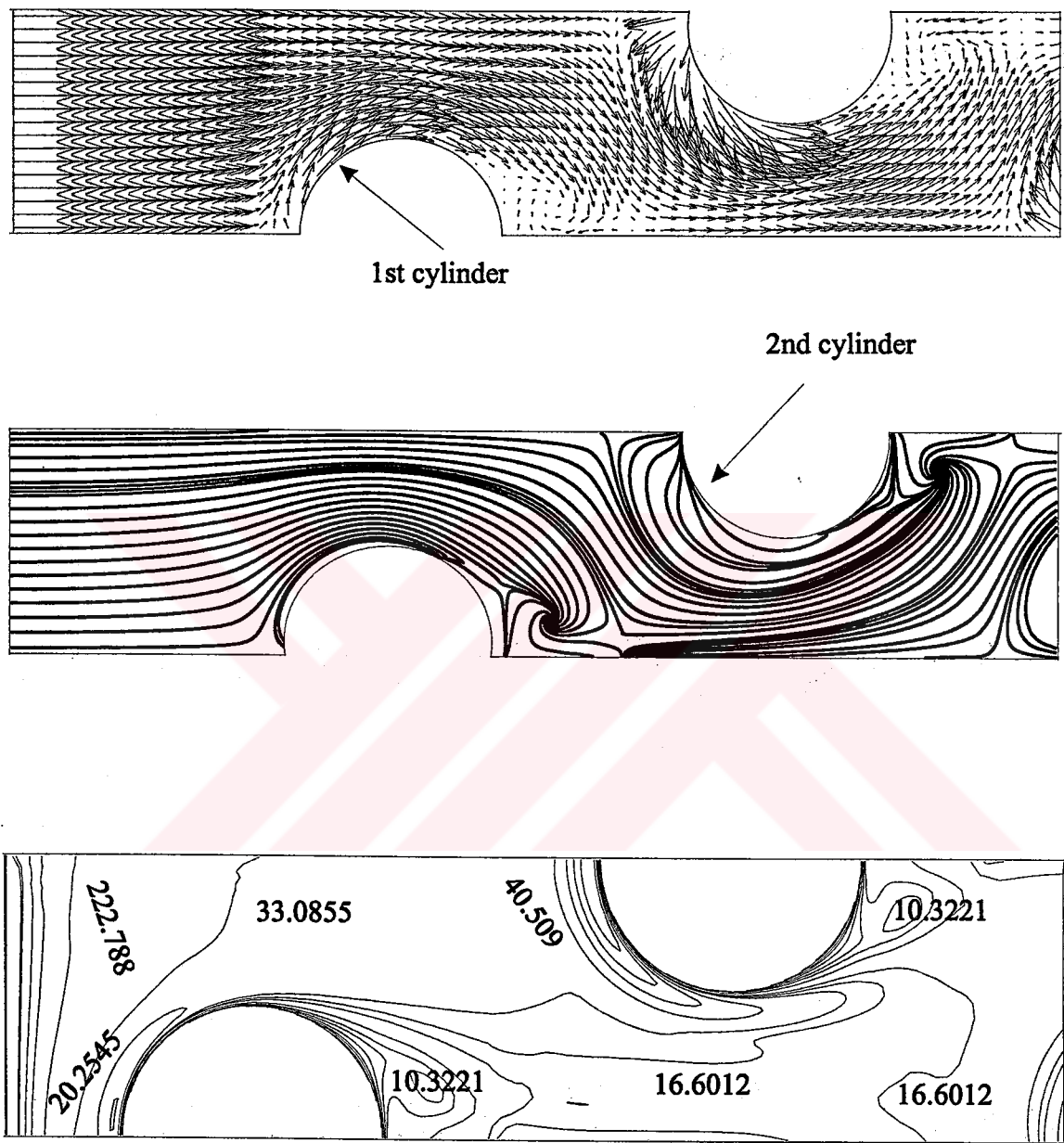


Figure 5.13c. Velocity vector, streamline and heat transfer data at $Z=Fs/12$ plane (very close to the fin surface) around the first and the second cylinders for the wavy fin configuration at $Re = 1000$, laminar model results

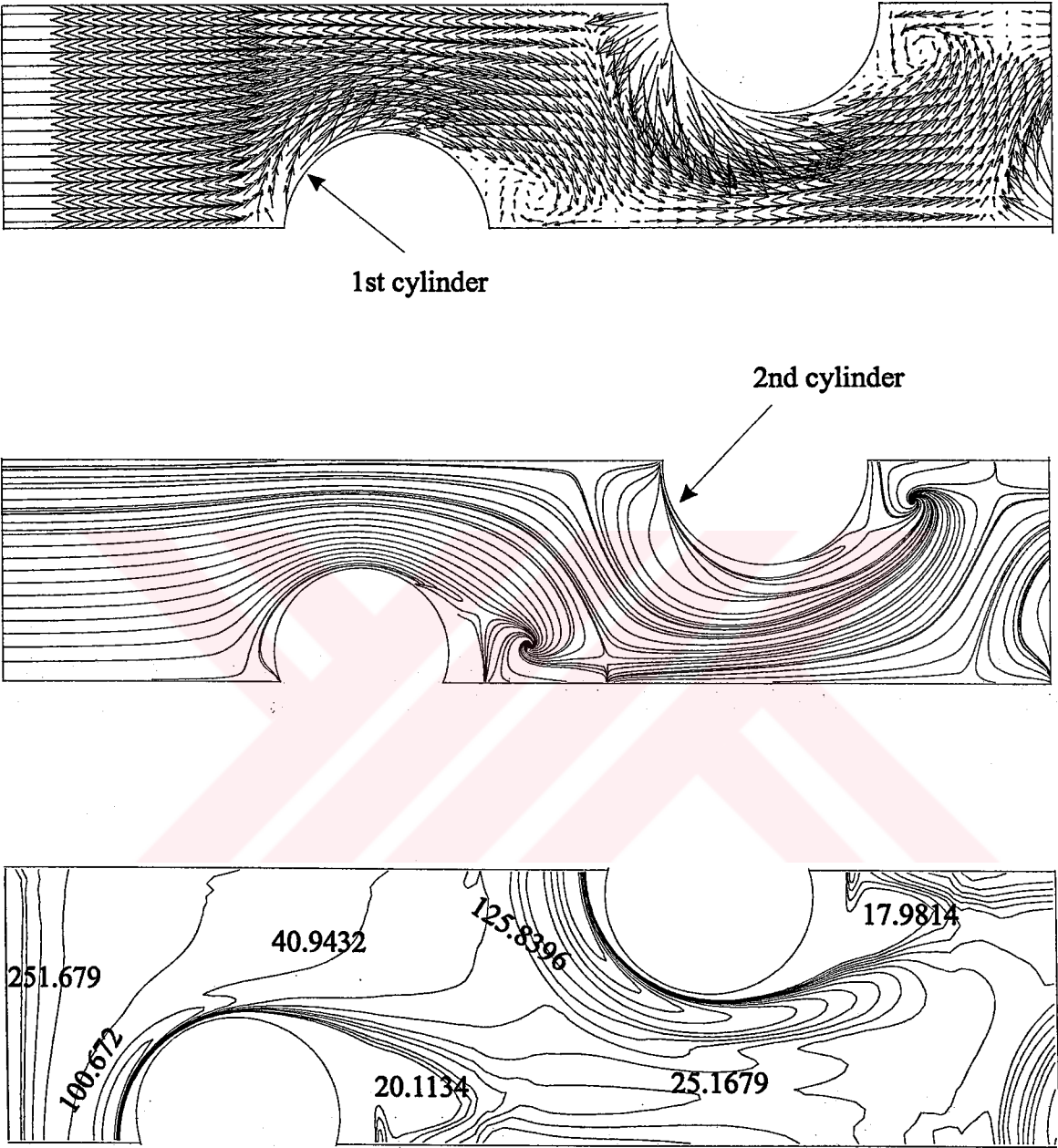


Figure 5.13d. Velocity vector, streamline and heat transfer data at $Z=Fs/12$ plane (very close to the fin surface) around the first and the second cylinders for the wavy fin configuration at $Re = 1500$, laminar model results

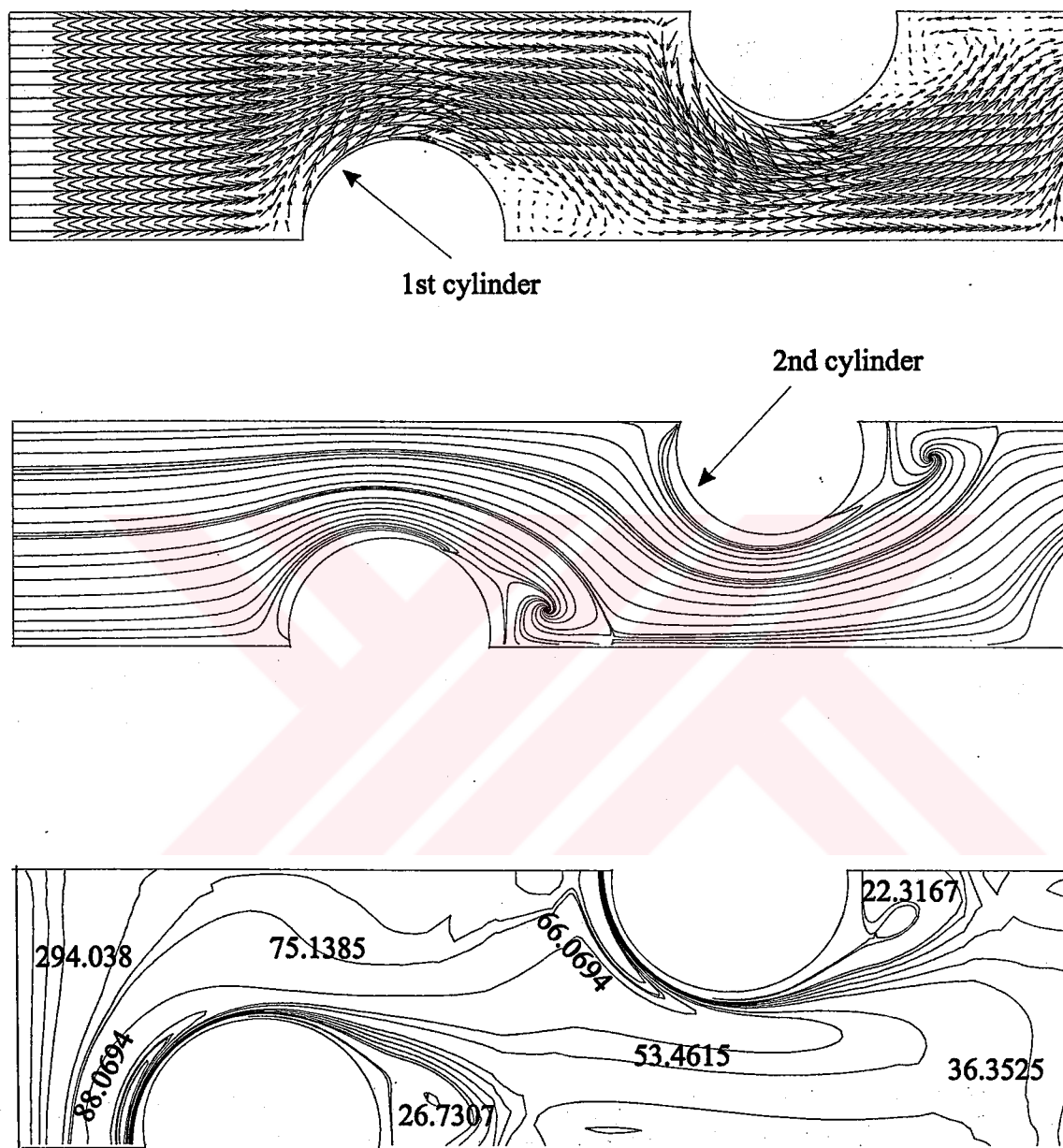


Figure 5.13e. Velocity vector, streamline and heat transfer data at $Z=Fs/12$ plane (very close to the fin surface) around the first and the second cylinders for the wavy fin configuration at $Re = 1500$, RNG k-e model results

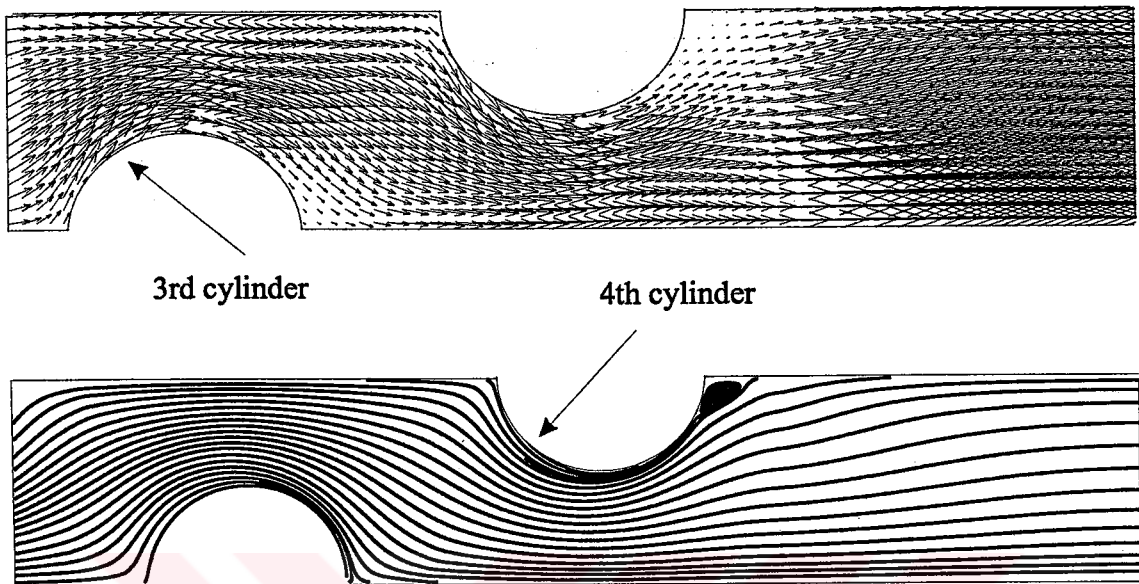


Figure 5.14a. Velocity vector and streamlines for the staggered layout, 4- row, wavy fin configuration around the third and the fourth cylinders at $Z=Fs/12$ plane at $Re=100$, laminar model simulation

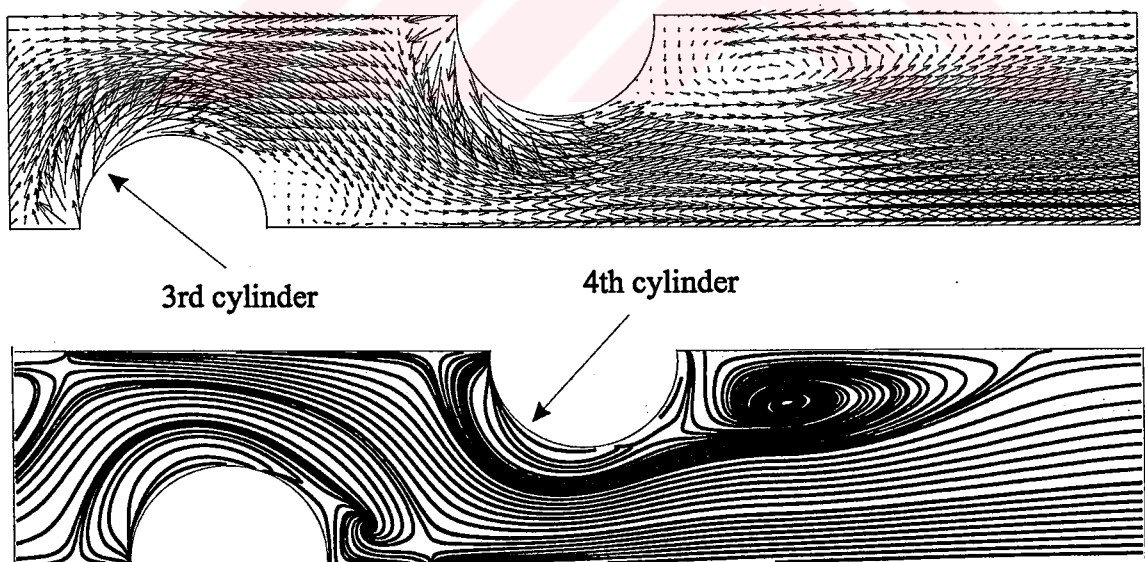


Figure 5.14b. Velocity vector and streamlines for the staggered layout, wavy fin configuration around the third and the fourth cylinders at $Z=Fs/12$ plane at $Re=500$, laminar model simulation

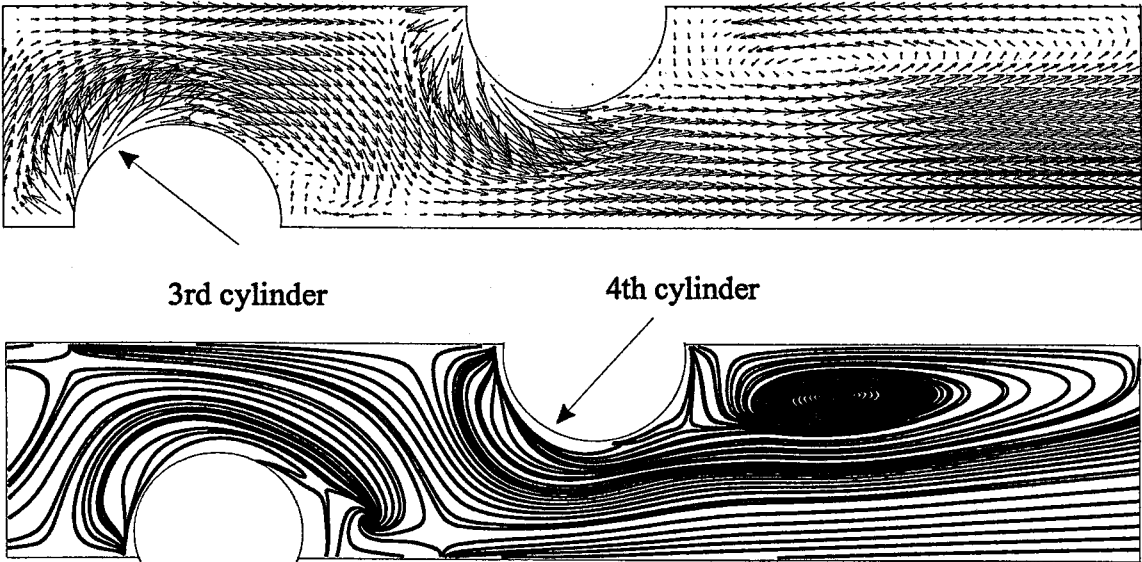


Figure 5.14c. Velocity vector and streamlines around the third and the fourth cylinders for the wavy fin configuration at $Z=Fs/12$ plane at $Re=1000$, laminar model simulation

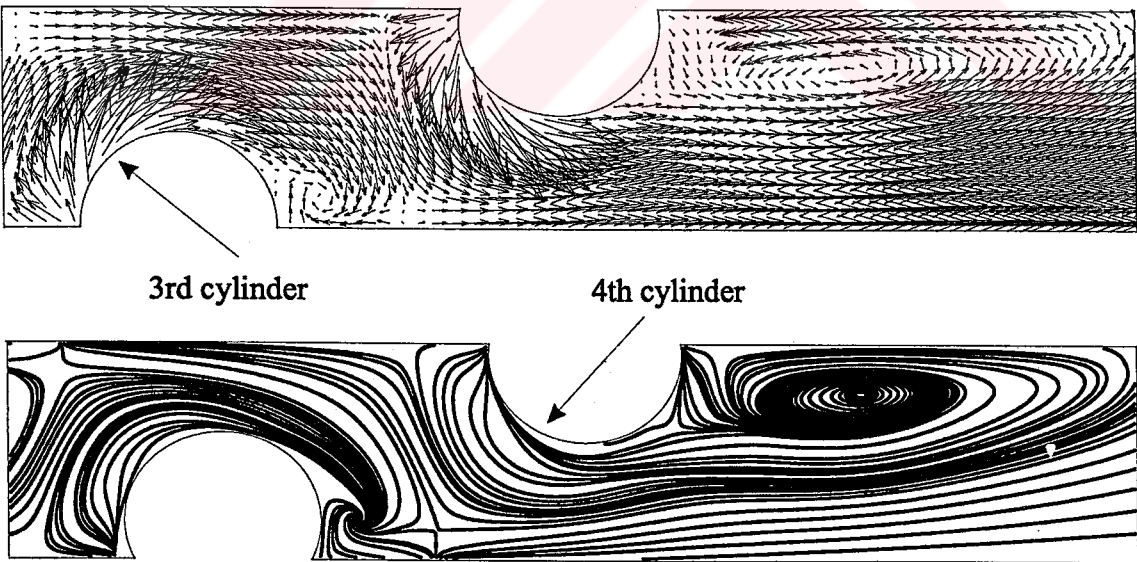


Figure 5.14d. Velocity vector and streamlines around the third and the fourth cylinders for the wavy fin configuration at $Z=Fs/12$ plane at $Re=1500$, laminar model simulation

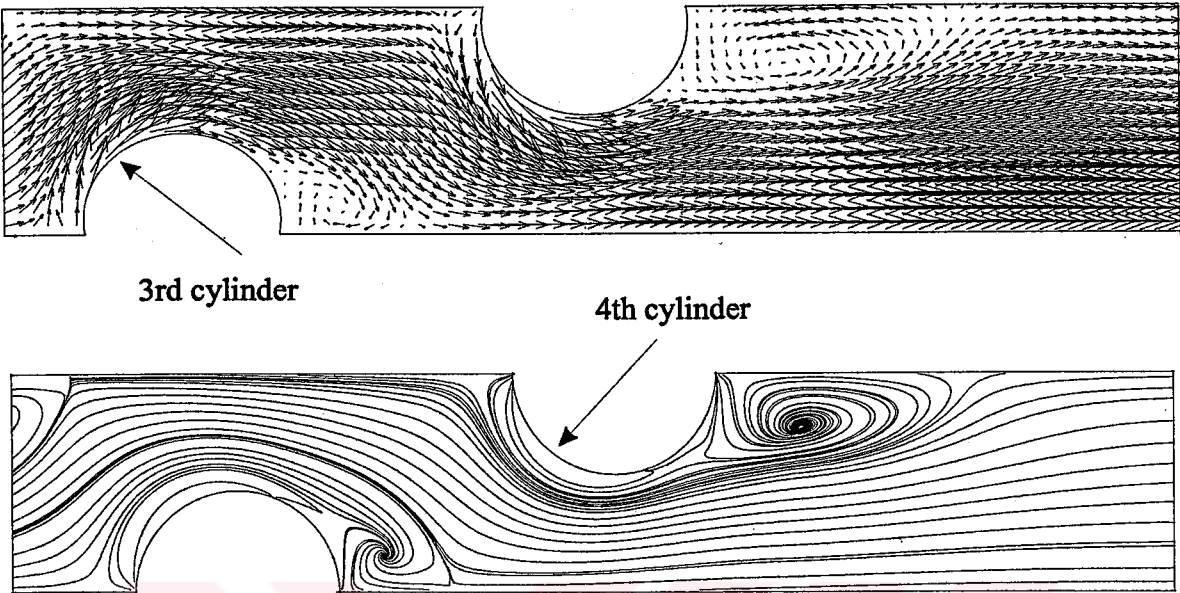


Figure 5.14e. Velocity vector and streamlines around the third and the fourth cylinders for the staggered layout, wavy fin configuration sat $Z=Fs/12$ plane at $Re=1500$, RNG k-e model simulation

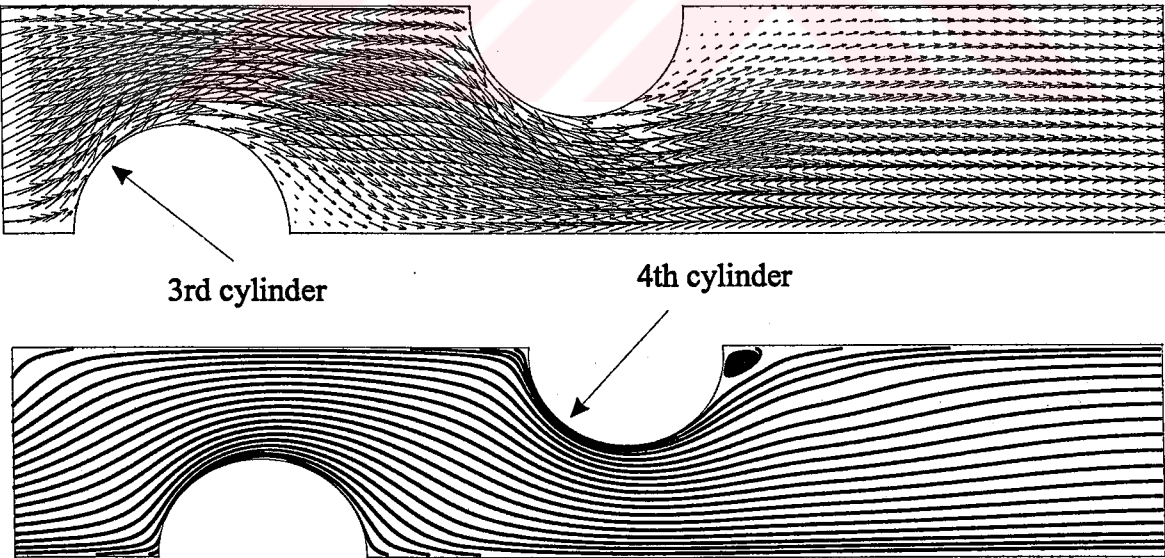


Figure 5.15a. Velocity vector and streamlines around the third and the fourth cylinders for the staggered layout, wavy fin configuration at $Z=Fs/2$ plane at $Re=100$, laminar model simulation

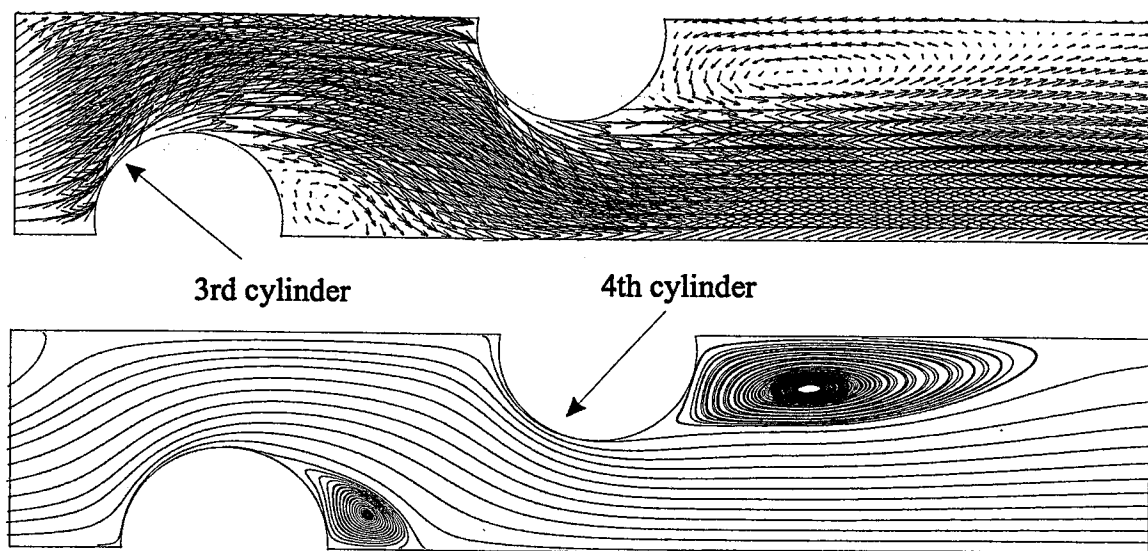


Figure 5.15b. Velocity vector and streamlines around the third and the fourth cylinders for the staggered layout, wavy fin configuration at $Z=Fs/2$ plane at $Re=500$, laminar model simulation

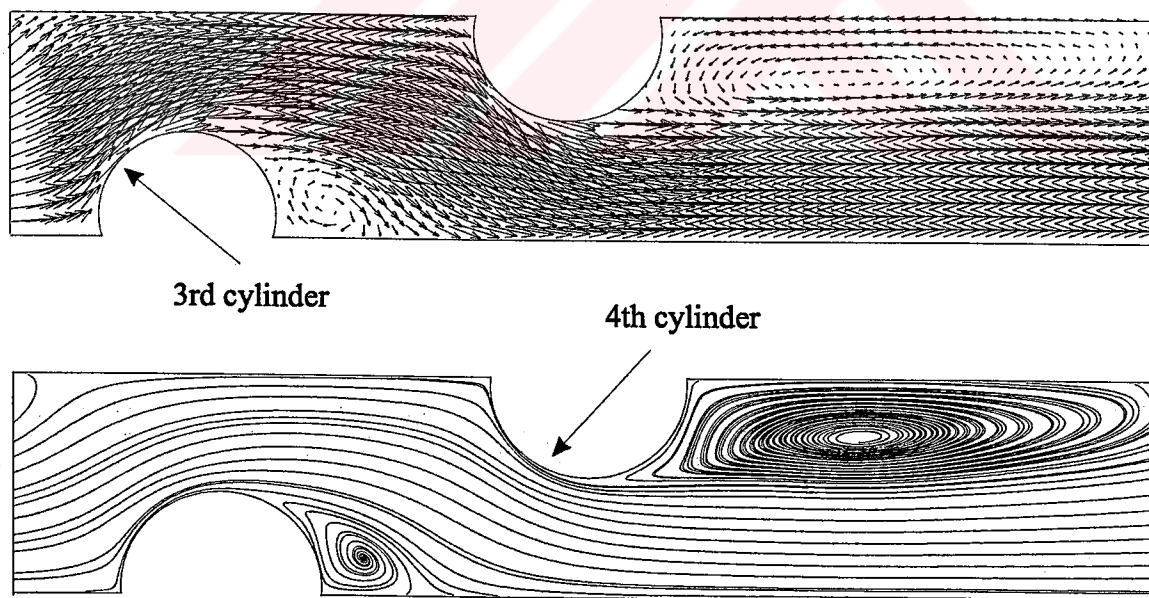


Figure 5.15c. Velocity vector and streamlines around the third and the fourth cylinders for the staggered layout, wavy fin configuration at $Z=Fs/2$ plane at $Re=1000$, laminar model simulation

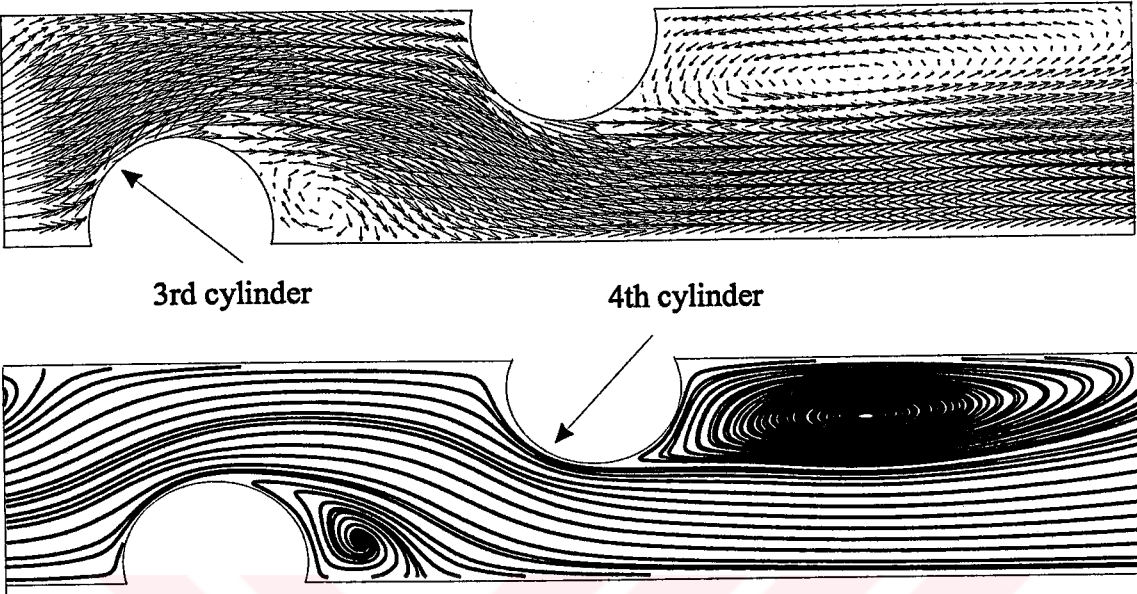


Figure 5.15d. Velocity vector and streamlines around the third and the fourth cylinders for the wavy fin configuration at $Z=Fs/2$ plane at $Re=1500$, laminar model simulation

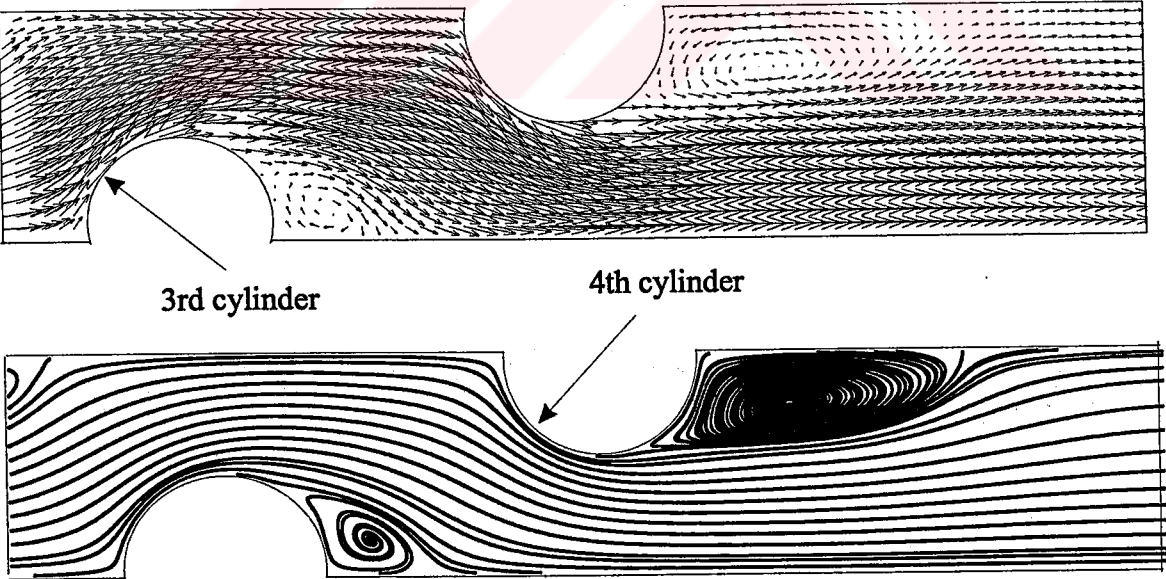


Figure 5.15e. Velocity vector and streamlines around the third and the fourth cylinders for the wavy fin configuration at $Z=Fs/2$ plane at $Re=1500$, RNG k-e model simulation

5.4. Effect of Selection of Different Wall Functions on the Heat Transfer and Pressure Drop in the Heat Exchanger Flow Passage

As it is discussed in Section 4.5, accurate representation of the flow in the near wall region determines successful predictions of wall- bounded turbulent flows. Wall functions are used to bridge the viscosity-affected region between the wall and the fully turbulent region. In FLUENT, three different wall functions are available: 1) Standard Wall Function, 2) Enhanced Wall Function, 3) Non- equilibrium Wall Function. Figure 5.16 and 5.17 show pressure drop coefficient, K , calculated by using Eq. 5.5, between the inlet and outlet sections of the heat exchanger as a function of Reynolds number obtained from Standard $k-\varepsilon$ and RNG $k-\varepsilon$ turbulence model simulations. It is clear that K decreases as the Reynolds number is increased. The Non- equilibrium wall function gives higher K values while the Enhanced wall function gives lower K values compared to the other wall functions for Standard $k-\varepsilon$ turbulence model simulation (Fig. 5.16). There is no significant difference between Standard and Enhanced wall functions on the determination of the K when $Re \geq 3000$ for RNG $k-\varepsilon$ turbulence model (Fig. 5.17). The same comments can be made for the heat transfer coefficient (Figs. 5.18 and 5.19). In Figs. 5.20 and 5.21, the calculated values for the friction factor (f) as a function of the Reynolds number are presented for the Standard $k-\varepsilon$ and RNG $k-\varepsilon$ turbulence models. The experimental data of Wang et al. (1997) are also included for a comparison. It is clearly seen in these figures that the wall function that best fits with the experimental data is the Enhanced wall function.

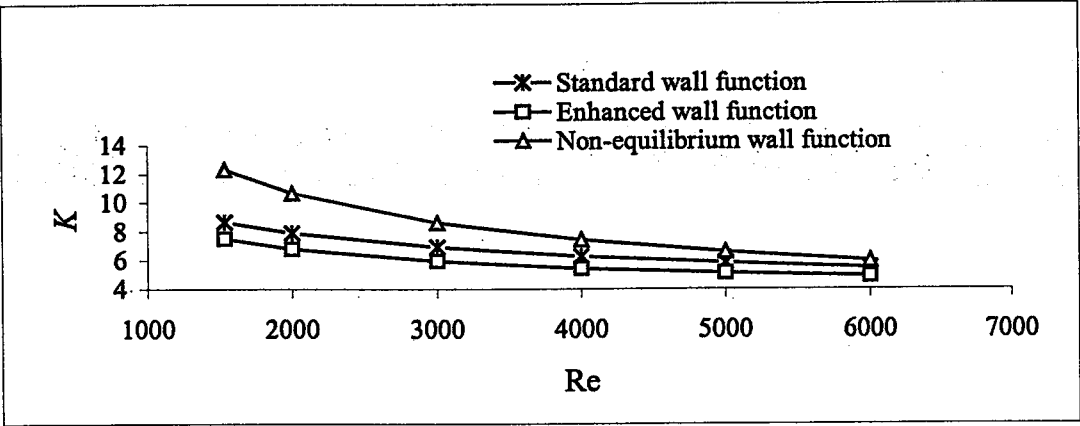


Figure 5.16. Pressure drop coefficient between the inlet and outlet sections of the heat exchanger as a function of Reynolds number obtained from Standard $k-\epsilon$ turbulence model simulation

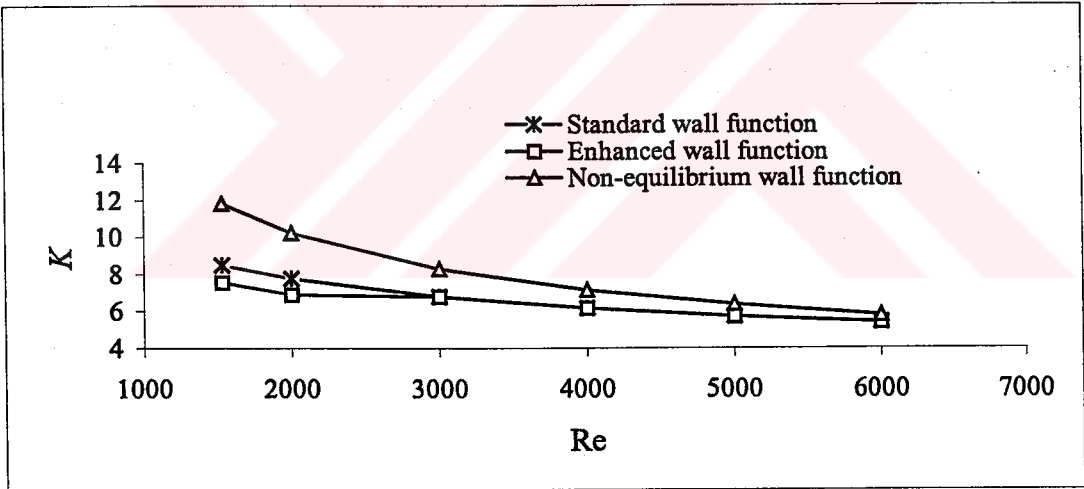


Figure 5.17. Pressure drop between the inlet and outlet sections of the heat exchanger as a function of Reynolds number obtained from RNG $k-\epsilon$ turbulence model simulation

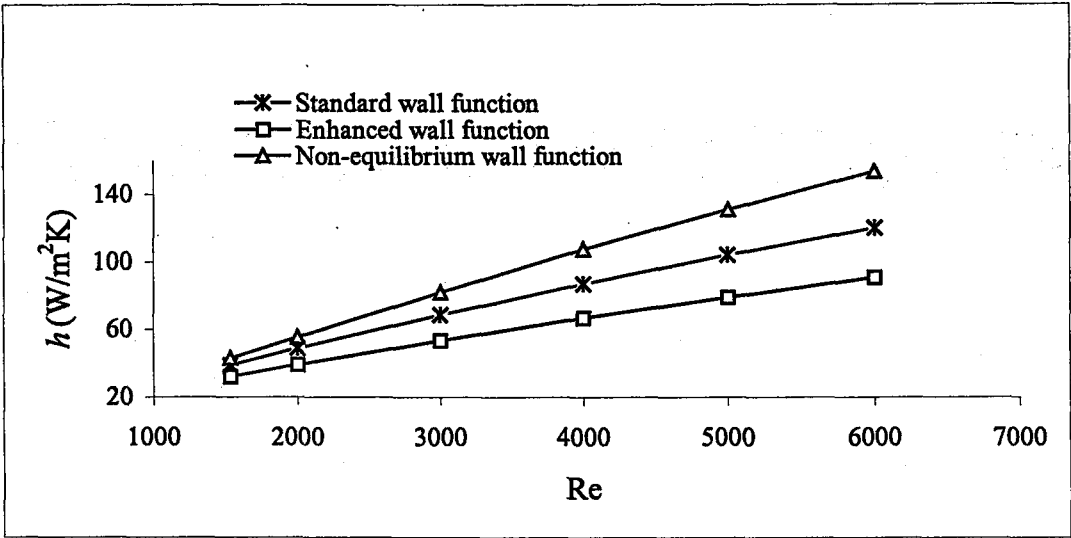


Figure 5.18. Heat transfer coefficient as a function of Reynolds number obtained from Standard $k-\epsilon$ turbulence model simulation

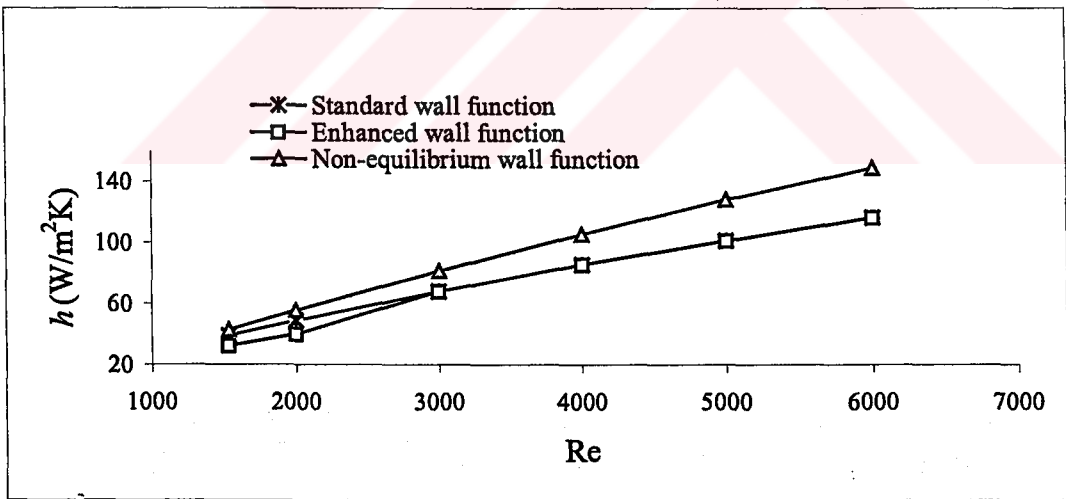


Figure 5.19. Heat transfer coefficient as a function of Reynolds number obtained from RNG $k-\epsilon$ turbulence model simulation

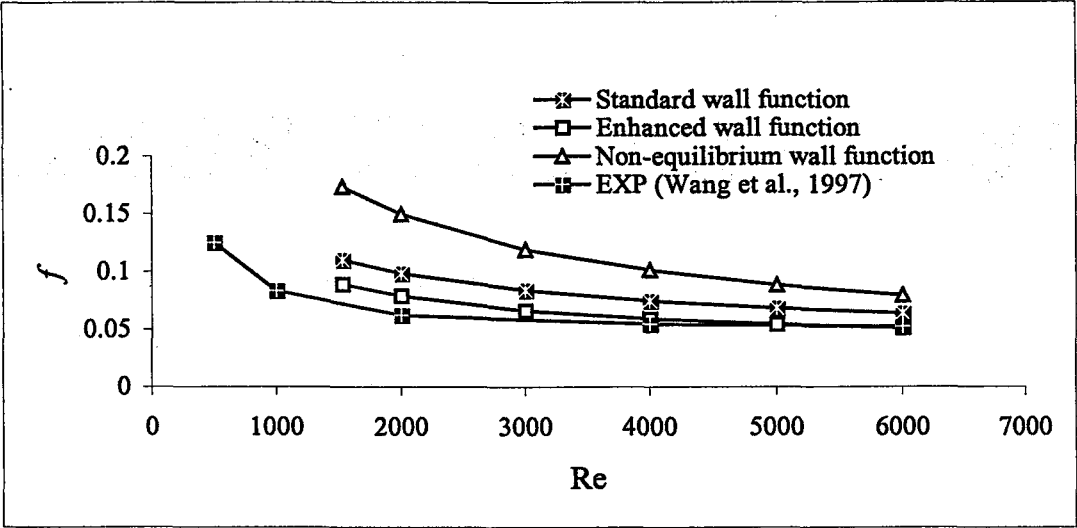


Figure 5.20. Friction factor as a function of Reynolds number obtained from Standard $k-\epsilon$ turbulence model simulation

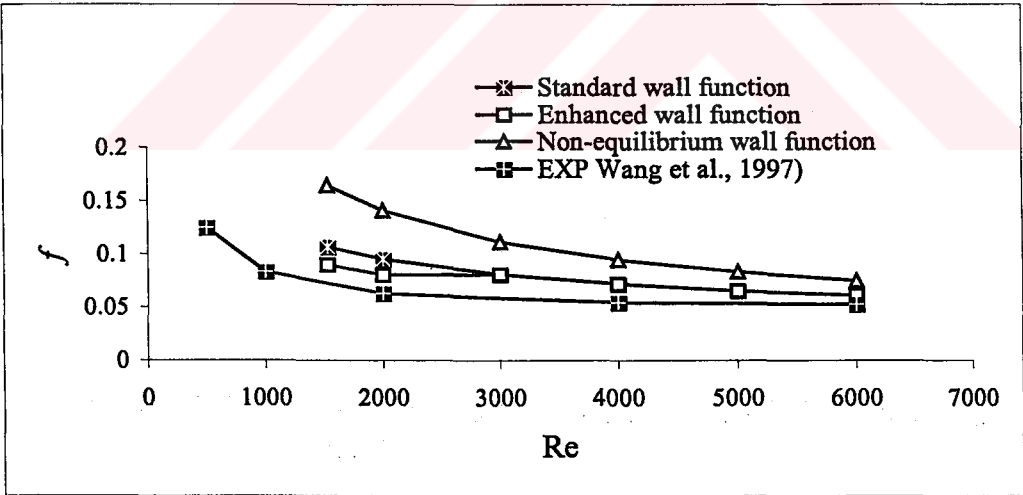
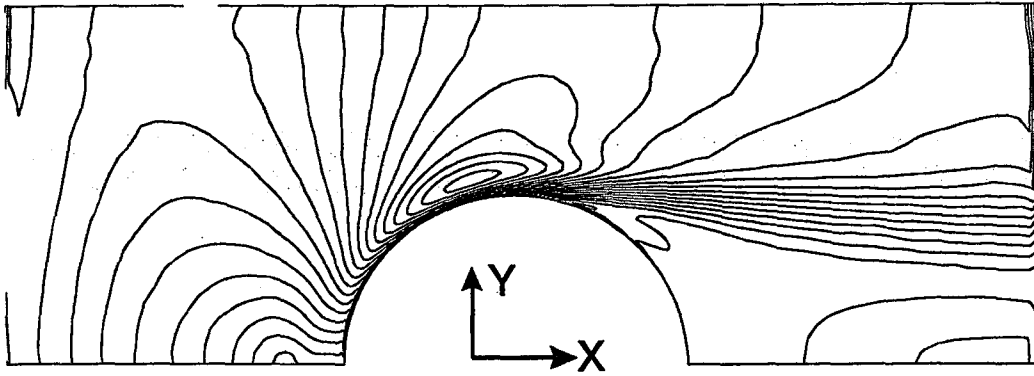


Figure 5.21. Friction factor as a function of Reynolds number obtained from RNG $k-\epsilon$ turbulence model simulation

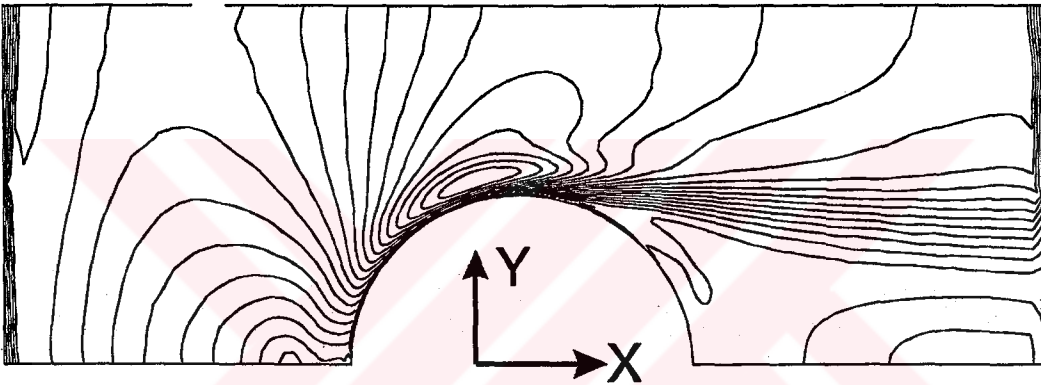
5.5. Turbulence Model Results

Having completed the test simulations regarding laminar flow assumption and 3-D steady state flow conditions, all governing geometrical parameters of the optimum heat exchanger have been determined and the present investigation has been extended to include real flow conditions and turbulence effects in the heat exchanger flow passage. In order to include the turbulence effects on the flow structure of the plate fin and tube heat exchangers, the turbulence model simulations have been carried out and the results are compared with each other for a single cylinder and plain fin configuration.

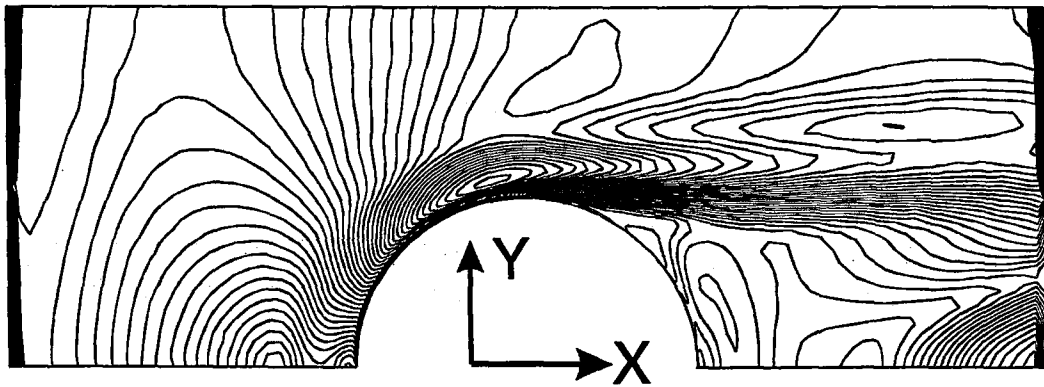
The development of the flow structure and the corresponding horseshoe vortex formation mechanism were further investigated for different turbulence model simulations for $F_s = 0.233$ at $Re = 4000$. Figure 5.22 shows time averaged vorticity contours around the tube surface in the $Z = 0$ plane. Both Standard $k-\epsilon$ (Fig. 5.22a) and RNG $k-\epsilon$ (Fig. 5.22b) turbulence models reproduce very similar horseshoe vortex mechanism in front of the cylinder. The LES (Fig. 5.22c) on the other hand reproduces more compact and stronger horseshoe vortex mechanism compared to the RANS equations based turbulence models. The location of the horseshoe vortex for the LES is also different than the other turbulence models: It moves further downstream around the cylinder surface resulting in better heat transfer performance in the upstream half of the tube as well as higher pressure rise in the wake of the cylinder.



(a)



(b)



(c)

Figure 5.22. Time averaged vorticity contours around the cylinder surface at $F_s = 0.233$, $Re = 4000$, in the $Z = 0$ plane, a) Standard $k-\epsilon$ model b) RNG $k-\epsilon$ model c) LES method

Figure 5.23 presents the time- averaged pressure distributions around the tube surface at varying Reynolds number for different laminar and turbulence model simulations for $F_s = 0.233$ and for the single cylinder configuration. It is seen that the variations in the pressure profiles for each simulation look similar and the C_p (as defined in the Eq. 5.3) value increases as the Reynolds number is increased. Maximum pressure coefficient value occurs in front of the cylinder due to the stagnation and sudden change exists in the calculated pressure profiles just upstream of the cylinder due to the existence of the horseshoe vortex. The horseshoe vortex system occurs as a continuation of developing boundary layers around and just upstream of the cylinder and the minimum C_p value associates with the location of the horseshoe vortex. The choice of numerical modeling, i.e. laminar or turbulence model is influential on the calculated base pressure values but its influence on the pressure distribution is not as significant as the fin spacing and Reynolds number as previously discussed in Fig. 5.4. At a fin spacing value of $F_s = 0.233$ for $Re = 1460$, both the Standard $k-\varepsilon$ and RNG $k-\varepsilon$ turbulence model simulations produce very similar pressure profiles with slightly different calculated time- averaged base pressure coefficient values of - 4.20 and - 4.24 respectively. On the other hand, much higher base pressure values are obtained from LES compared to the laminar and the RANS equations based turbulence model simulations at the same fin spacing and Reynolds number. For LES, at $F_s = 0.233$, the calculated time- averaged base pressure coefficient values for $Re = 1460$, 3000, and 5000 are found to be -2.78, - 2.45, and -2.12 respectively. Higher time- averaged pressure coefficient values obtained from LES cause the tube to experience less pressure forces compared to laminar and RANS equations based time averaged turbulence model simulations.

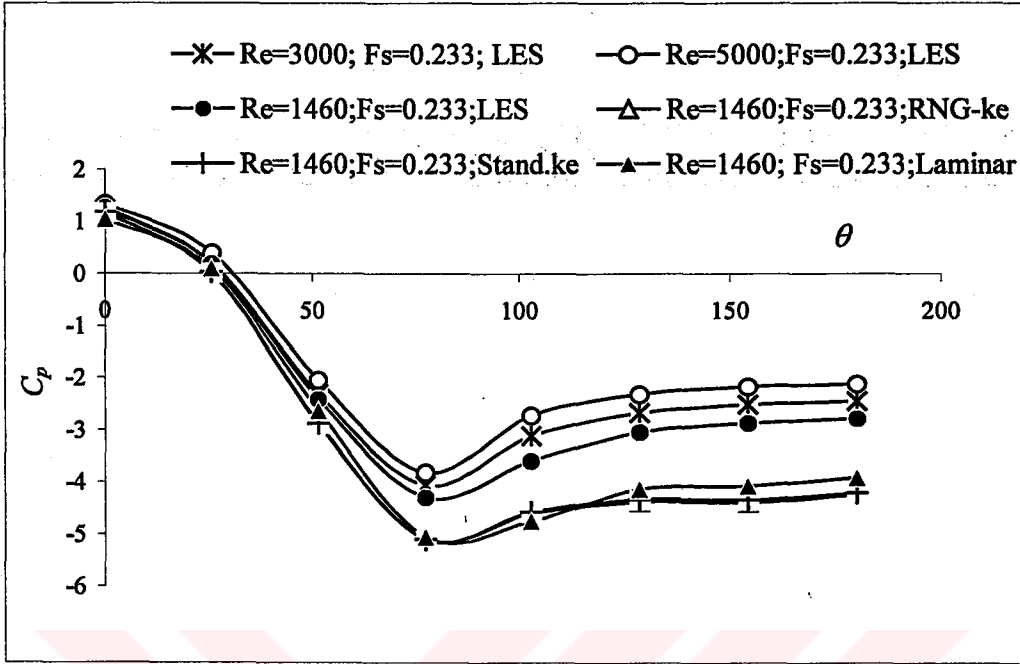
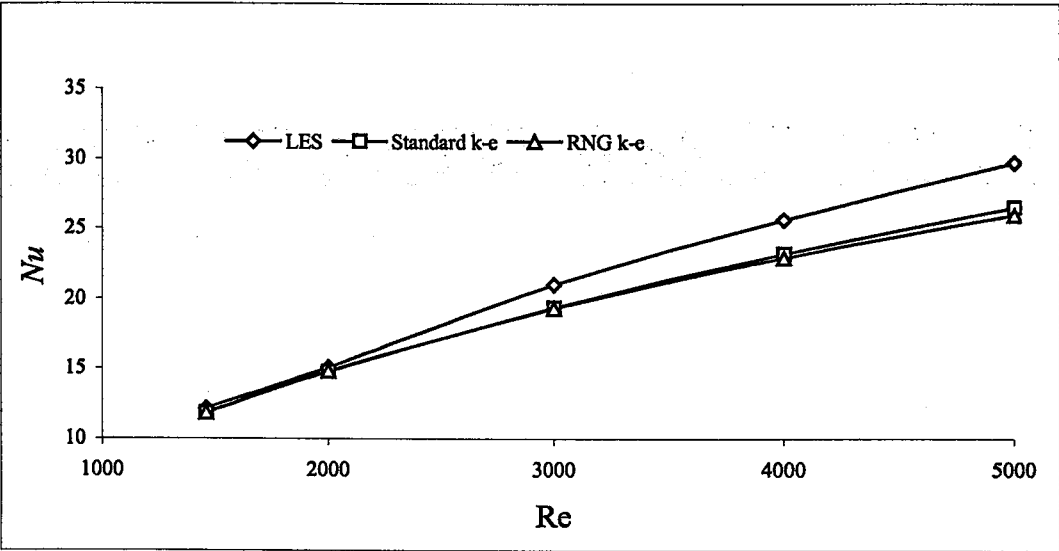


Figure 5.23. Time averaged local pressure distribution around the single cylinder, plain fin configuration, obtained from turbulence model simulations

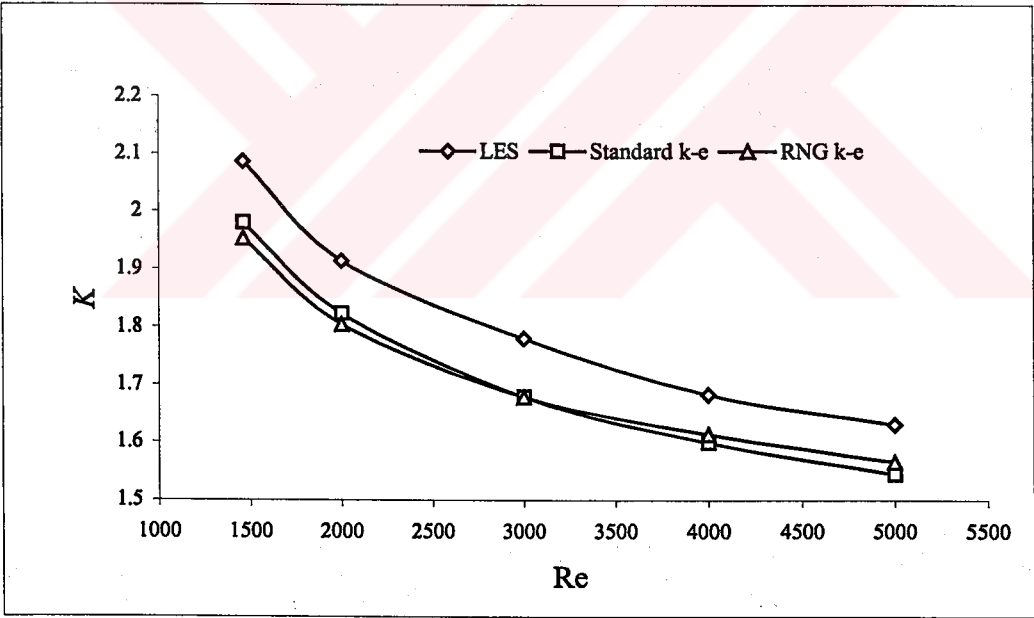
Figure 5.24 shows relation between average Nusselt number on the fin and the tube surfaces and pressure drop coefficient across the heat exchanger against Reynolds number obtained for different turbulence model simulations at a fin spacing value of $F_s = 0.233$, for the single cylinder configuration. The area-weighted averaged values of the convective heat transfer coefficient, h , are obtained from computation of the summation of surface integrals on the fin and the tube surfaces, by using Eq. (5.4). Then the average Nusselt number based on the fin spacing as the characteristic length is calculated from Eq. (5.8). As stated by Kakac and Liu (1997), pressure drop across a compact heat exchanger aids in determining pumping power or fan work input necessary to maintain the flow of fluids flowing in the system. Therefore in the present study the pressure drop coefficient across the heat exchanger is expressed as the pressure drop coefficient (K) (Eq. 5.5). It is clearly seen from Fig.5.24a,b that, higher calculated values of the Nusselt number and pressure drop coefficient are obtained from LES compared to those obtained from RANS equations

based turbulence models. This is due to the fact that more compact and stronger horseshoe vortices obtained from LES compared to the RANS equations based time averaged turbulence models cause better heat transfer characteristics and also higher pressure drop across the heat exchanger passage. Although identical boundary conditions are imposed for both RANS equations based turbulence models and the LES calculations, the differences of calculated values of pressure drop coefficient and Nusselt number are mainly caused by differences in the turbulence modeling.





(a)

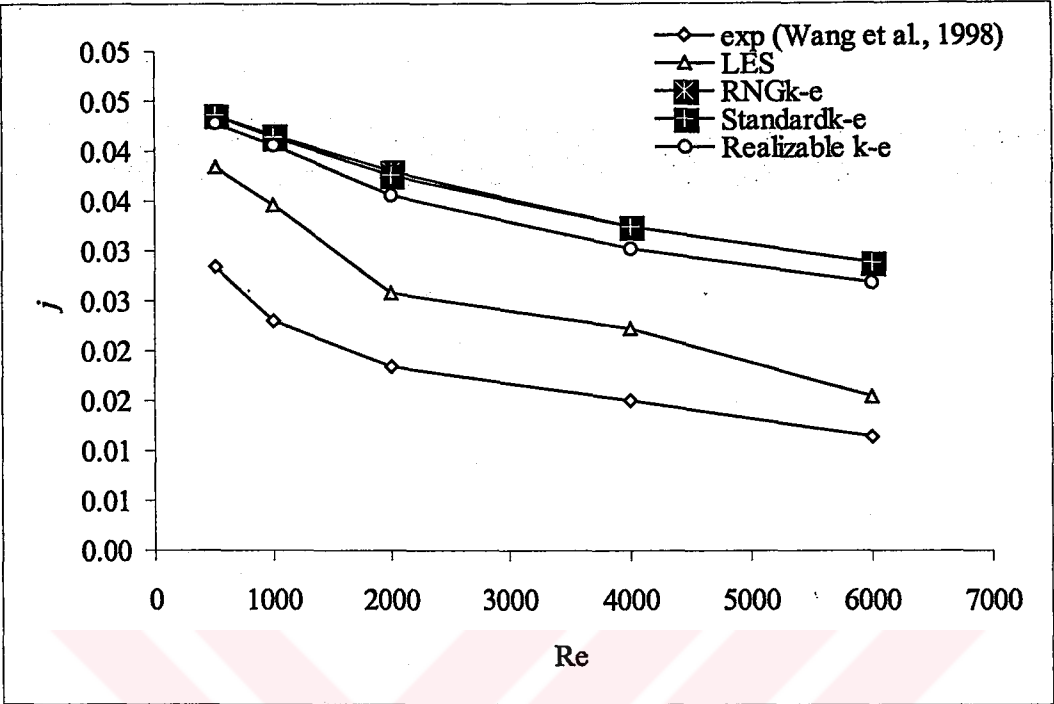


(b)

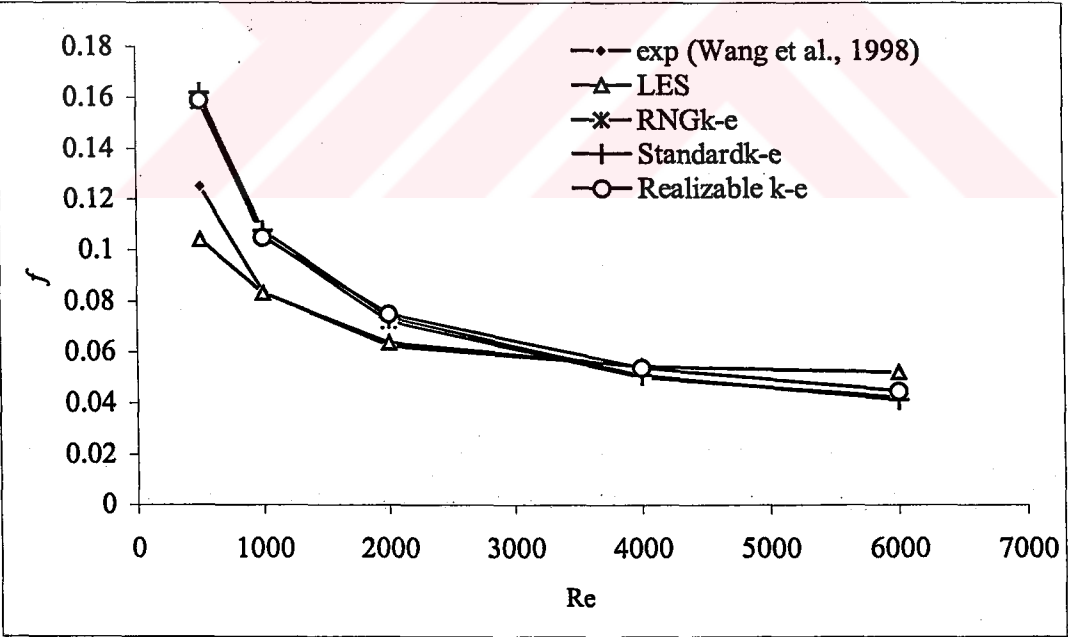
Figure 5.24. Average Nusselt number and pressure drop coefficient across the heat exchanger as a function of Reynolds number for $F_s = 0.233$ (Misag 147, 2003).

Figure 5.25 shows the calculated values of the j factor, (Eq.5.2), against Reynolds number based on outer tube diameter considering different turbulence model simulations for four cylinder and wavy fin configuration for $F_s = 0.233$. Experimental data of Wang et al. (1998) is also included for comparison. As it is seen from Figure 5.25a, j factor decreases with increasing Reynolds number. All numerical model simulation results overestimate the experimental results. The calculated values for the j factor obtained from RANS equations based turbulence model simulations are very close to each other. On the other hand, LES simulation gives better agreement with the experimental data compared to the RANS equations based turbulence model simulations. This is due to the fact that RANS equations based turbulence models do not consider the velocity changes with time. Another fact is the turbulence effects are not considered in the experiments. However even the LES simulation result is not very close to the experimental result. Then it is concluded that LES method should be further improved. If the three turbulence model simulations are compared with each other, Realizable $k-\varepsilon$ model gives better results for the calculated values of the j factor than the Standard $k-\varepsilon$ and the RNG $k-\varepsilon$ turbulence models. The values obtained from RANS equations based turbulence model simulations are higher than those obtained from LES simulations.

Figure 5.25b represents calculated values and experimental data of f factor, (Eq. 5.1), against Reynolds number based on outer tube diameter considering different turbulence model simulations for four cylinder and wavy fin configuration for $F_s = 0.233$. Experimental data of Wang et al. (1998) is also included for comparison. As the Reynolds number is increased, f decreases. All results are in good agreement with the experimental data. The turbulence model that best fits with the experimental data is the LES method as it is clearly seen from Fig. 5.25b.



(a)

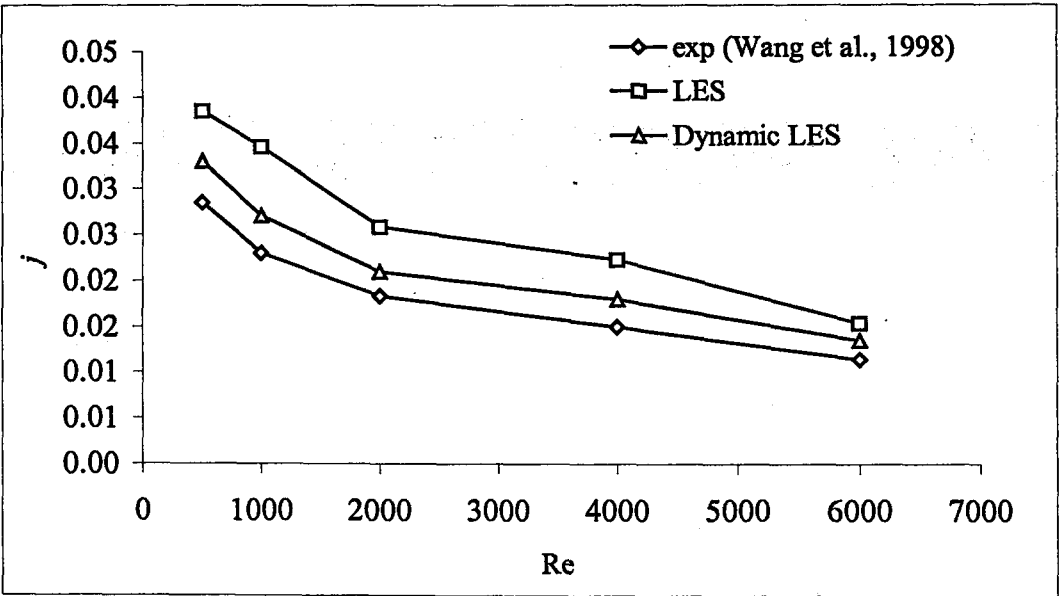


(b)

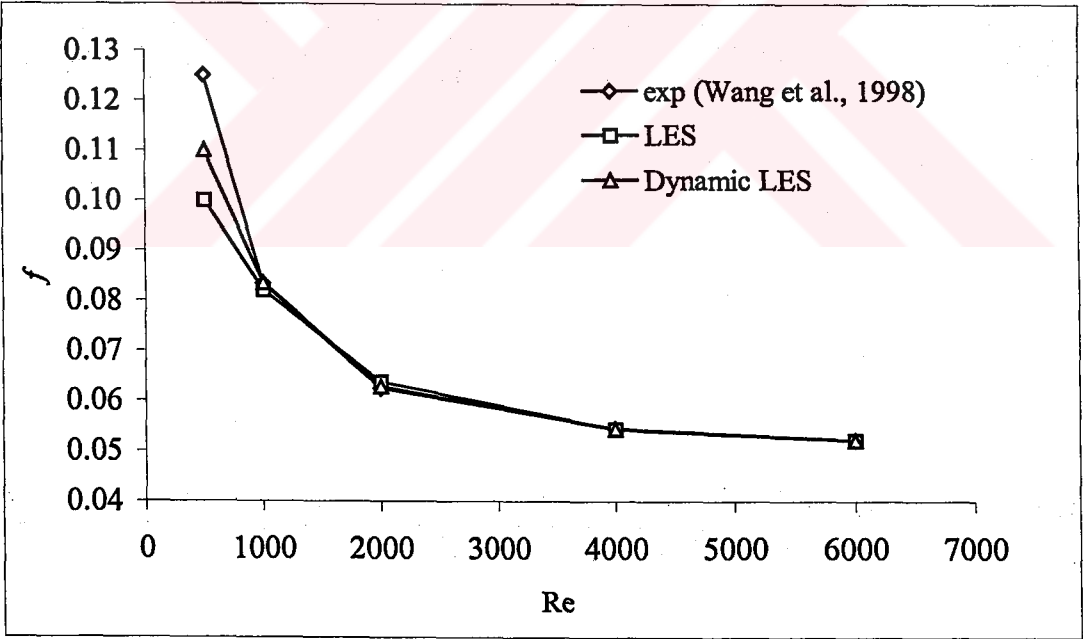
Figure 5.25. Colburn factor (a) and Friction factor (b) as a function of Reynolds number obtained for different turbulence model simulations for wavy, four cylinder, staggered layout heat exchanger geometry for $F_s = 0.233$ (Misag 147, 2003).

In order to obtain better agreement with the experimental data it is thought that the Standard LES method should be further improved to include the dynamic effects as explained before. Therefore the Dynamic LES method is applied to the flow under consideration and the calculated results for j and f factors are obtained as shown in Fig. 5.26. As it is clearly seen from the figure, inclusion of space and time variation of the Smagorinsky constant to the standard model gives better results that best fits with the experimental data compared to the standard model.





(a)



(b)

Figure 5.26. Colburn factor (a) and Friction factor (b) as a function of Reynolds number obtained for Standard LES and Dynamic LES turbulence model simulations for wavy, four cylinder, staggered layout heat exchanger geometry for $F_s = 0.233$ (Misag 147, 2003).

Figure 5.27 and 5.28 show streamlines on the fin surface obtained for the optimum heat exchanger model, around the second and the third cylinder obtained from Standard LES and Dynamic LES model simulations, respectively, at non-dimensional time steps of $\tau = 4.5, 9$ and 22 . At $\tau = 4.5$, a focus of separation rotating in the counter clockwise and clock wise sense behind the second and the third cylinder, respectively, can be clearly seen for both models (Figs. 5.27a and 5.28a). The focus behind the second and the third cylinder disappears at $\tau = 9$, and at $\tau = 22$, respectively for Standard LES model (Fig. 5.27b and c). Instead, the streamlines are feed into a nodal point of separation located on a close region of the cylinder surface. On the other hand, the foci behind the second and the third cylinder remain nearly in the same position as at $\tau = 9$ for the Dynamic LES simulation (Fig. 5.28b). At $\tau = 22$, the foci behind the third and the second cylinder are about to disappear and a lot of singular points are formed behind the second and the third cylinders (Fig. 5.28c).

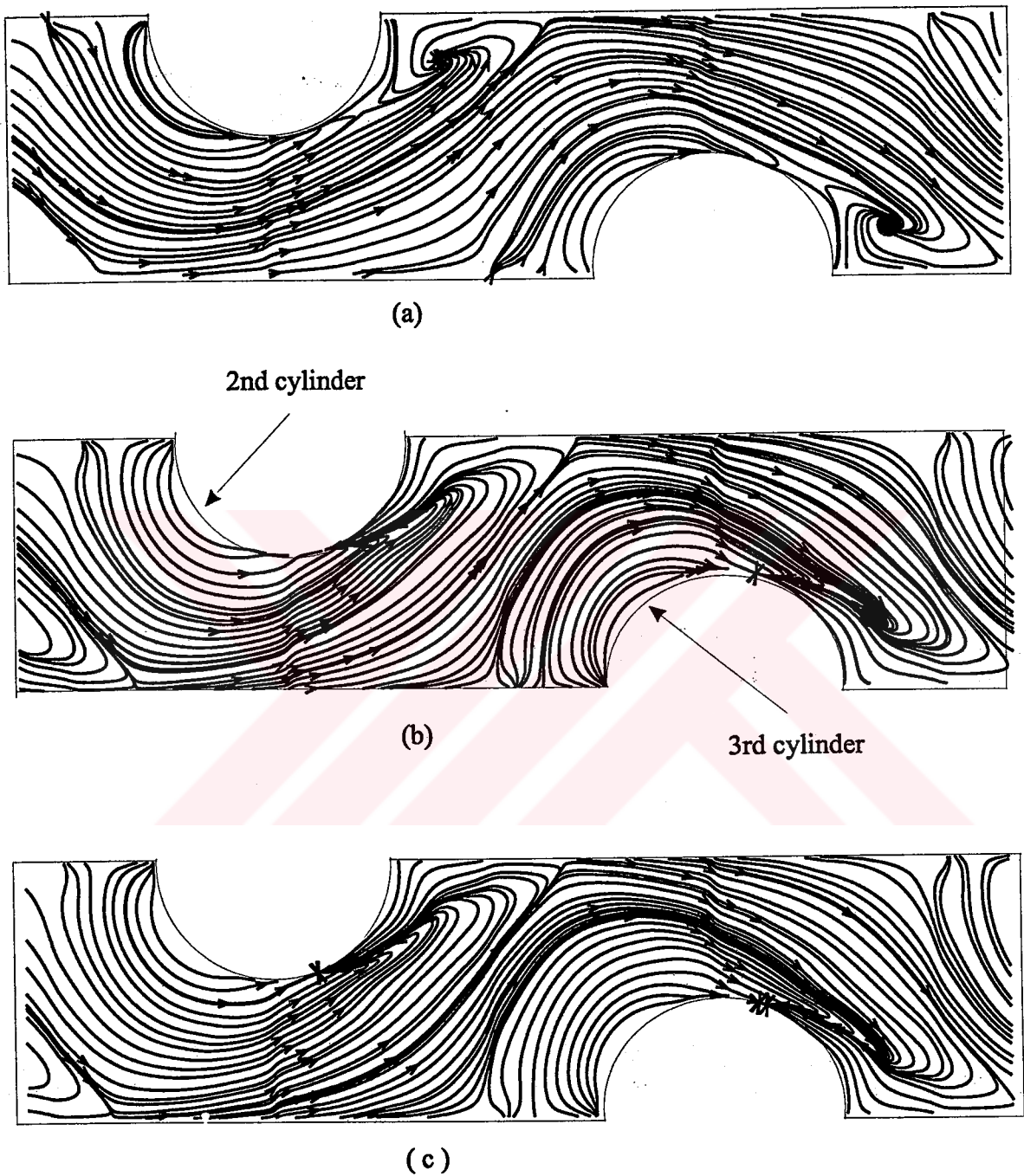


Figure 5.27. Streamlines on the fin surface around the second and the third cylinder obtained from standard LES simulations for the staggered layout wavy fin configuration at non- dimensional time steps of a) 4.5, b)9, c) 22.

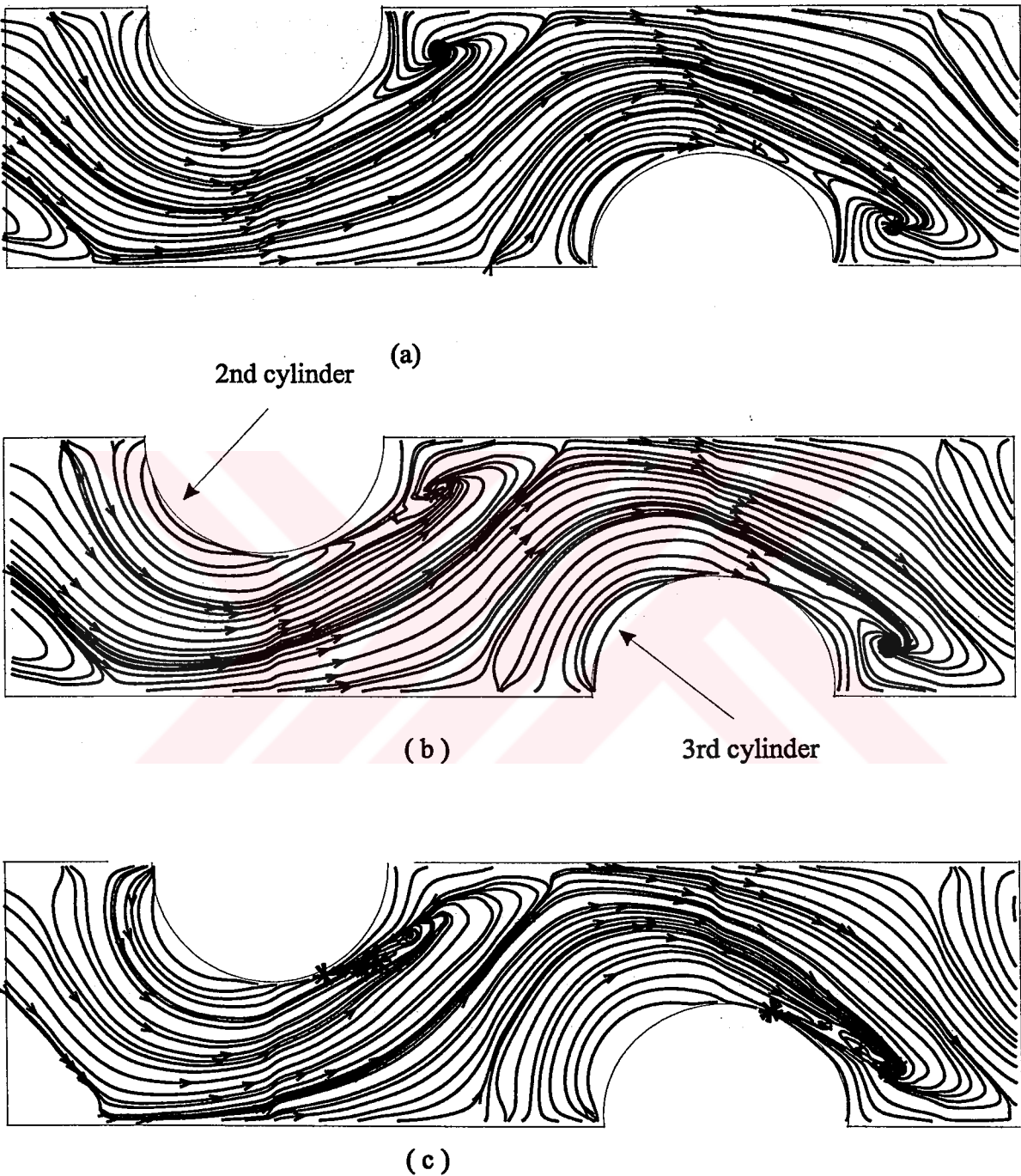


Figure 5.28. Streamlines on the fin surface around the second and the third cylinder obtained from dynamic LES simulations for the staggered layout wavy fin configuration at non- dimensional time steps of a) 4.5, b)9,c) 22.

Figure 5.29 and 5.30 show heat transfer coefficients around the second and the third cylinder on the fin surface for the 4 cylinder, staggered layout, wavy fin configuration obtained for Standard LES and Dynamic LES method simulations, respectively. It is found that higher heat transfer coefficients are obtained from the Dynamic LES method compared with the Standard LES method.



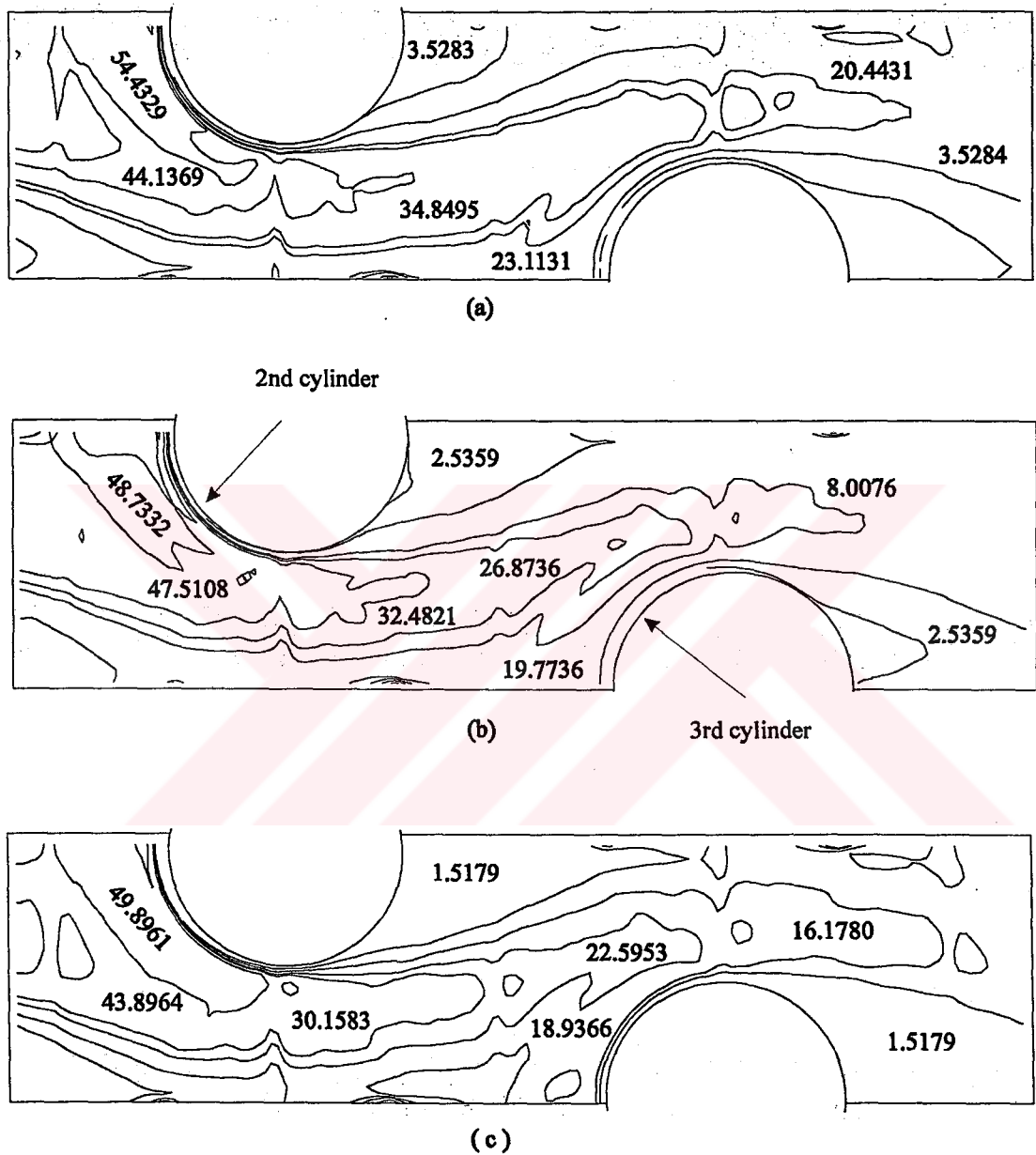
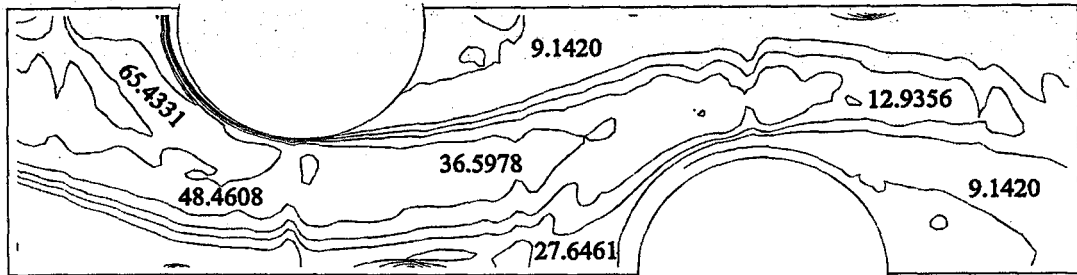
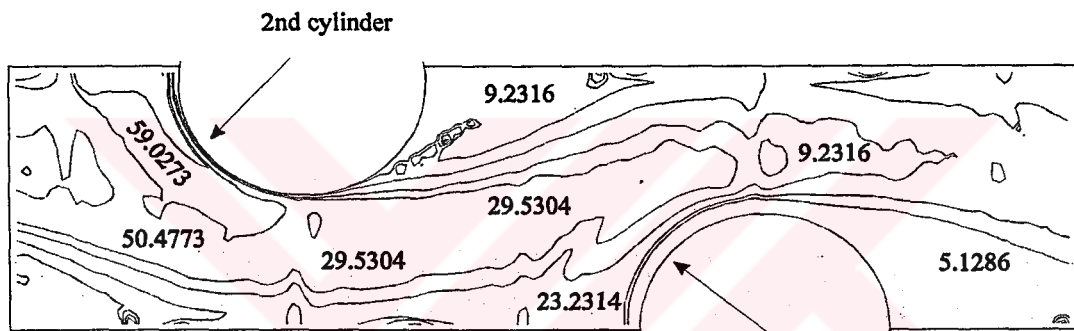


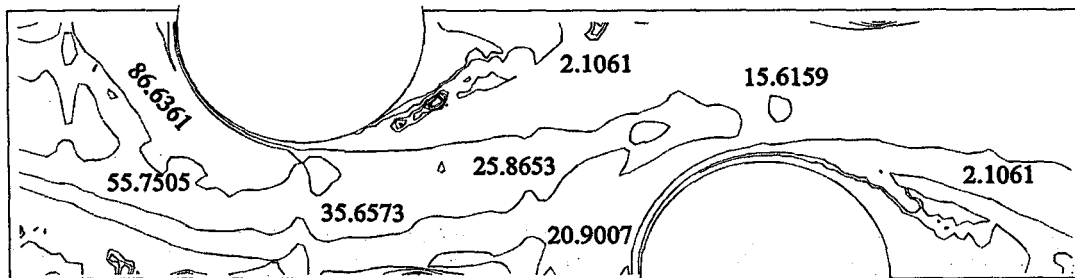
Figure 5.29. Heat transfer coefficients on the fin surface around the second and the third cylinder obtained for standard LES simulations for the staggered layout, wavy fin configuration at non-dimensional time steps of a) $\tau = 4.5$, b) $\tau = 9$, c) $\tau = 22$



(a)



(b)



(c)

Figure 5.30. Heat transfer coefficients on the fin surface around the second and the third cylinder obtained for Dynamic LES simulations for the staggered layout, wavy fin configuration at non-dimensional time steps of a) $\tau = 4.5$, b) $\tau = 9$, c) $\tau = 22$

Figure 5.31 and 5.32 show contours of vorticity magnitude at the center line axis of the flow passage obtained from Dynamic LES method simulations for the staggered layout, 4 cylinder, plain and wavy fin configuration, respectively at $Z = F_s / 2$ plane at non- dimensional time steps of $\tau = 20, 40$, and 80 . It is found that the vorticities are more elongated and widely scattered on the fin surface in case of wavy fin configuration. At the same time higher values of maximum vorticities are obtained for the wavy fin configuration as shown in Table 5.2.

Table 5.2. Minimum and maximum vorticity magnitude obtained from Dynamic LES method simulations for the staggered layout, 4 cylinder, plain and wavy fin configurations.

Non Dimensional Time Step	Plain Fin Configuration	Wavy Fin Configuration
$\tau = 20$	Min: 1.17 s^{-1} Max: $6.73 \times 10^4 \text{ s}^{-1}$	Min: 1.63 s^{-1} Max: $7.15 \times 10^4 \text{ s}^{-1}$
$\tau = 40$	Min: 1.74 s^{-1} Max: $7.45 \times 10^4 \text{ s}^{-1}$	Min: 3.00 s^{-1} Max: $7.65 \times 10^4 \text{ s}^{-1}$
$\tau = 80$	Min: 0.80 s^{-1} Max: $8.27 \times 10^4 \text{ s}^{-1}$	Min: 2.57 s^{-1} Max: $9.08 \times 10^4 \text{ s}^{-1}$

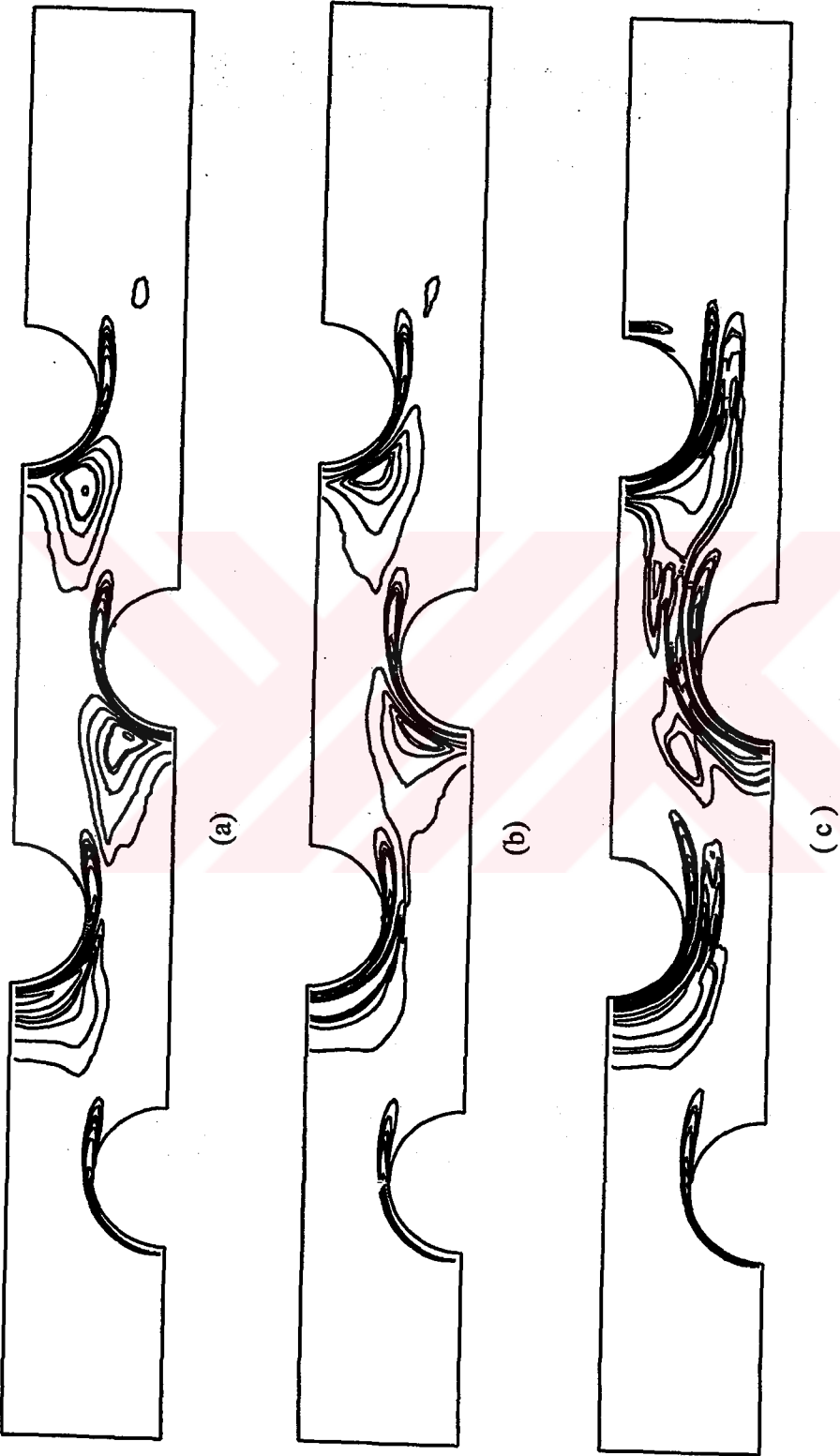


Figure 5.31. Contours of vorticity magnitude at the center line axis of the flow passage ($Z=Fs/2$) for staggered layout, 4 cylinder, plain fin configuration, at non dimensional time steps of a)20, b)40, and c)80. The DynamicLES method simulation results

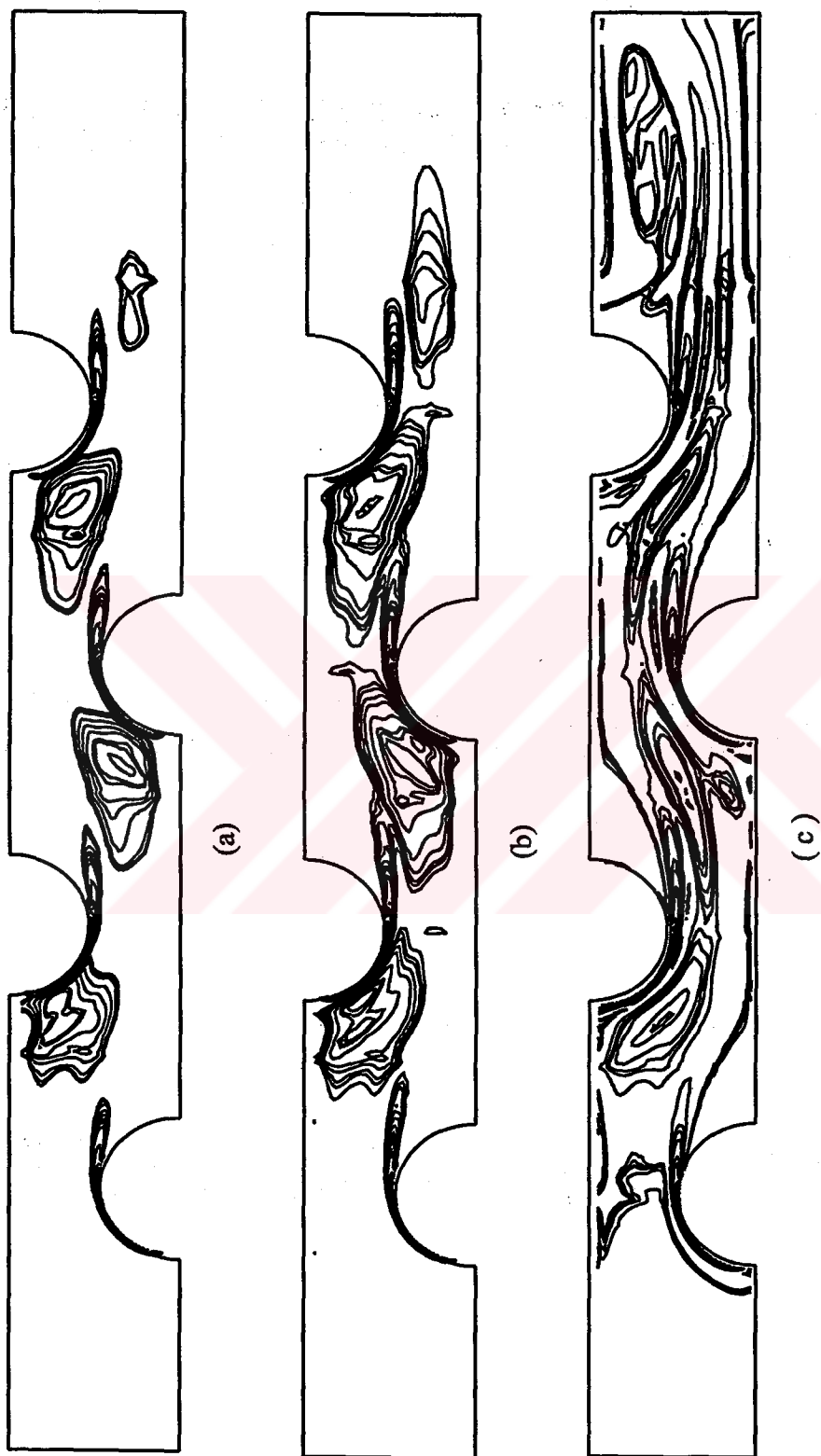


Figure 5.32. Contours of vorticity magnitude at the center line axis of the flow passage ($Z=Fs/2$) for staggered layout, 4 cylinder, wavy fin configuration, at non dimensional time steps of a) 20, b) 40, and c) 80. The Dynamic LES method simulation results

5.6. Laminar Model Transient Simulation Results

In the present study, the evolution of the horseshoe vortex mechanism and the vortices at the rear of the tube have been further investigated by the transient modelling because such mechanisms affect both heat transfer rate and pressure drop in the flow passages and they can not be predicted well with the steady state modelling that ignores the flow unsteadiness and the transient behaviour of the boundary layer development. For this reason, 3-D time dependent sample simulations have been carried out to capture the flow behavior around the tube surfaces and to accurately predict the corresponding heat transfer and pressure drop data in the flow passage of plate fin- and- tube heat exchangers.

The non- dimensional time step is expressed as the product of time averaged streamwise velocity component at the inflow boundary and time elapsed divided by the fin spacing as follows:

$$\tau = \frac{Ut}{F_s} \quad (5.6)$$

Where U is the time averaged streamwise velocity at the inlet, t is the time and the F_s is the fin spacing. For the time dependent simulations, the non dimensional time is chosen as a value of 0.004 and 0.01 for the single row configuration and multi row configuration, respectively.

Figure 5.33 shows instantaneous vorticity contours at $Z = 0.05$ plane for $F_s = 0.365$ and $Re = 1460$ at non-dimensional time steps of $\tau = 2, 6, 17$ and 52. Three vortices are observed at $\tau = 2$: A horseshoe vortex located in front of the tube, another vortex located after the separation point in the rear side of the tube, and a small vortex located near the wake region (Fig. 5.33a). As the flow approaches the tube, complex flow behavior results in the formation of horseshoe vortex in the forward part of the tube. The vortex located in the rear side of the tube which occurs after the separation point is of horseshoe type and caused by focusing of streamlines emerging from a saddle point located on the fin surface near the tube and the fin

junction as will be described in detail later. At $\tau = 2$, the numerical value of the maximum vorticity has a value of $2.11 \times 10^3 \text{ s}^{-1}$. At $\tau = 6$, the horseshoe vortex becomes stronger and both vortices at the rear of the tube grow in size. The vortex near the wake region splits the vortex in the rear side of the tube into two parts and pushes it to displace in the downstream direction while it remains attached to the tube wall (Fig. 5.33b). The numerical value of the maximum vorticity is obtained as $2.29 \times 10^3 \text{ s}^{-1}$ at $\tau = 6$. At $\tau = 17$, an outer horseshoe vortex appears in the neighborhood of the inner horseshoe vortex in front of the tube. Vortices are washing away into the wake interacting with each other can also be seen. The vortex near the wake region begins to detach from the tube wall moving further downstream in the wake region (Fig. 5.33c). The numerical value of the maximum vorticity is obtained as $7.32 \times 10^2 \text{ s}^{-1}$ at $\tau = 17$. At $\tau = 52$, the inner and the outer horseshoe vortices move slightly further downstream of the tube. A number of small vortices shed from the apex of the tube spread into the downstream of the tube and interact with each other in the wake. (Fig. 5.33d). The numerical value of the maximum vorticity is obtained again as $7.32 \times 10^2 \text{ s}^{-1}$ at $\tau = 52$.

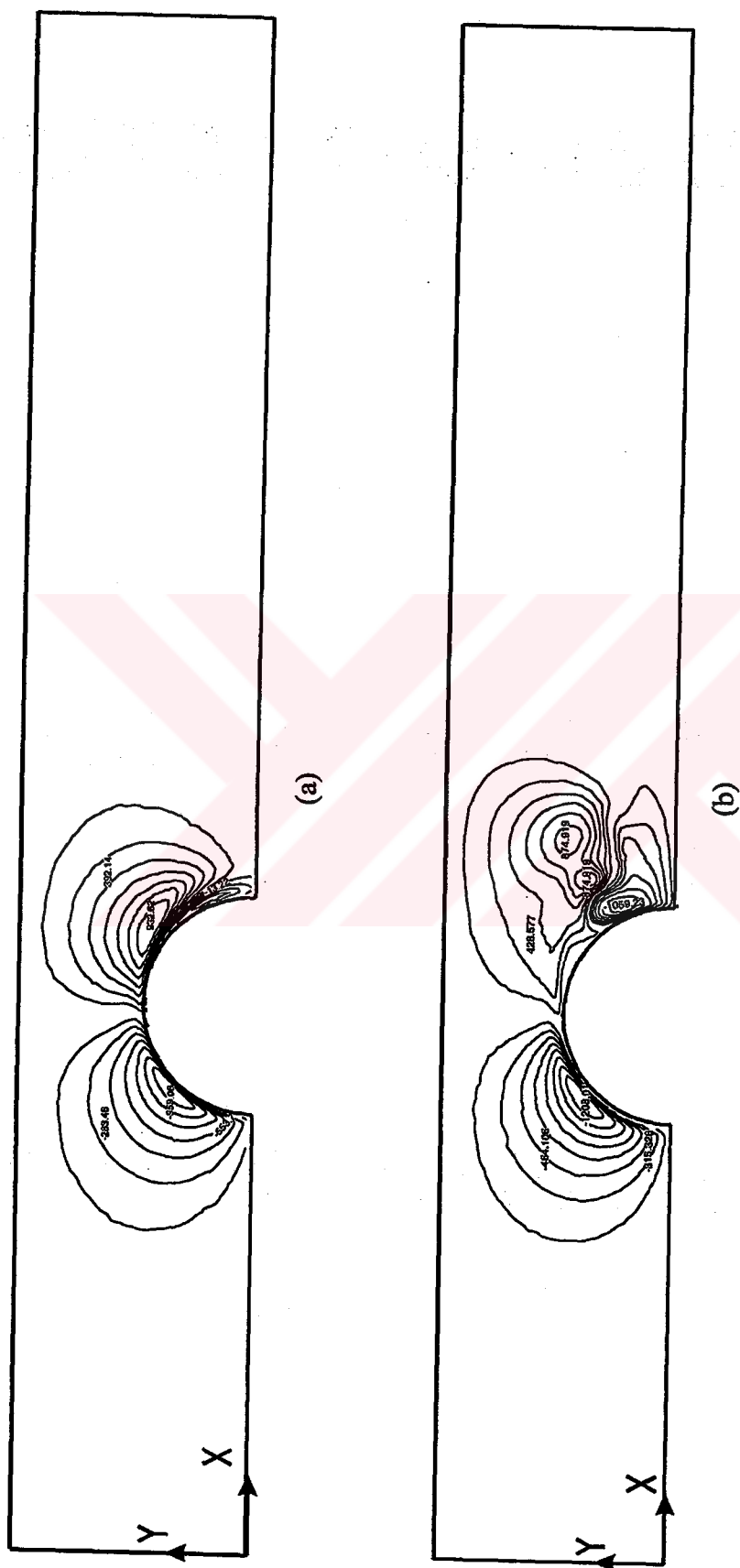


Figure 5.33. Instantaneous vorticity contours for transient simulations of the single cylinder configuration at different non-dimensional times, τ ; a) $\tau = 2$; b) $\tau = 6$

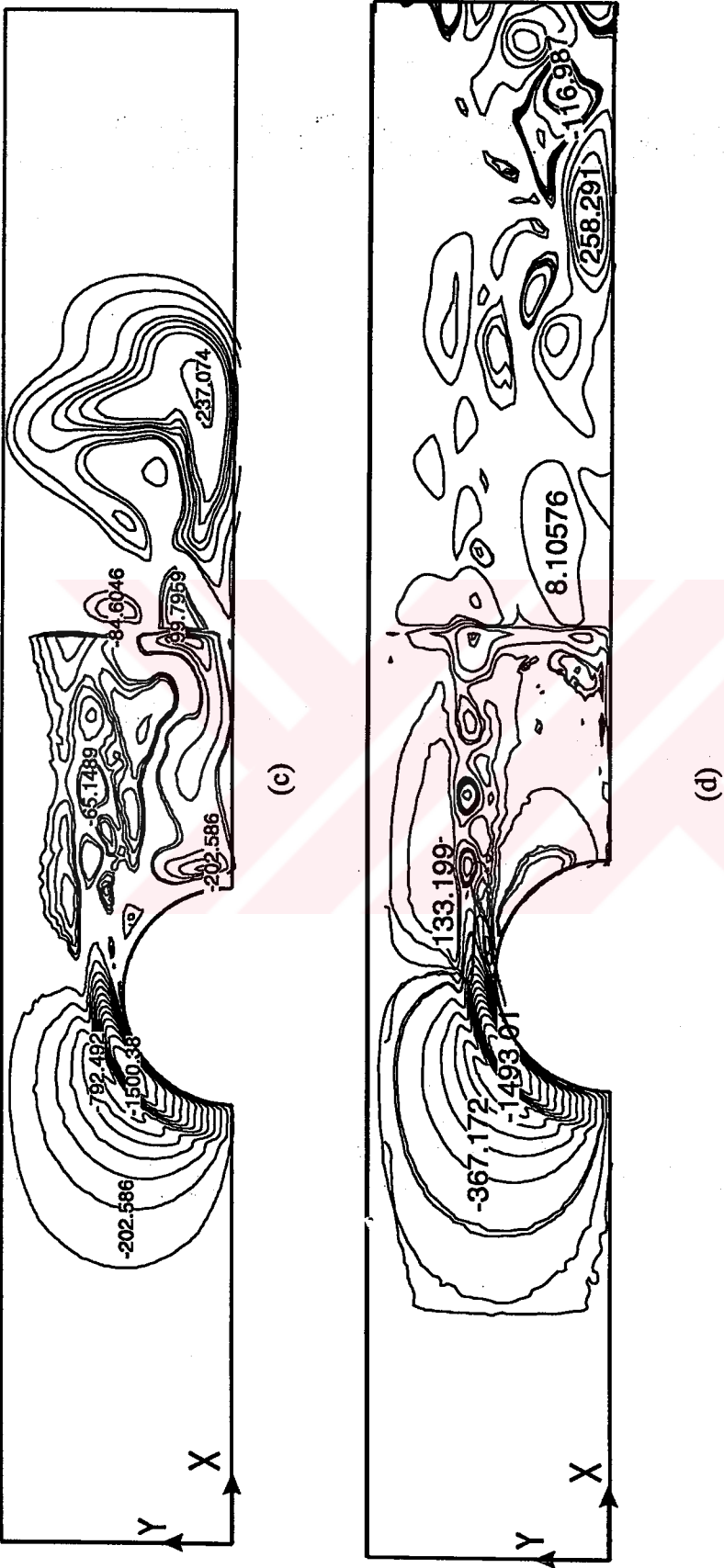


Figure 5.33. (continued) Instantaneous vorticity contours for transient simulations of the single cylinder configuration at different non-dimensional times, τ ; c) $\tau = 17$; d) $\tau = 52$

The time evolution of the flow structure in the flow passage is further investigated in terms of singular points. In the present study, singular points are found throughout the flow domain such as around the tube, on the fin, and at various locations in the flow passage. The singular points are also shown by Tsai and Sheu (1998) around the tube and on the fin surfaces. In order to capture local flow details around the tube surface the theta-cut planes are created as shown in Fig. 5.34. Each plane is numbered due to the angle theta between this plane and the x -axis. The angle theta is measured from forward stagnation point of the tube and changes between 0 and 180 degrees both for the single row case for each tube for the multi row case,

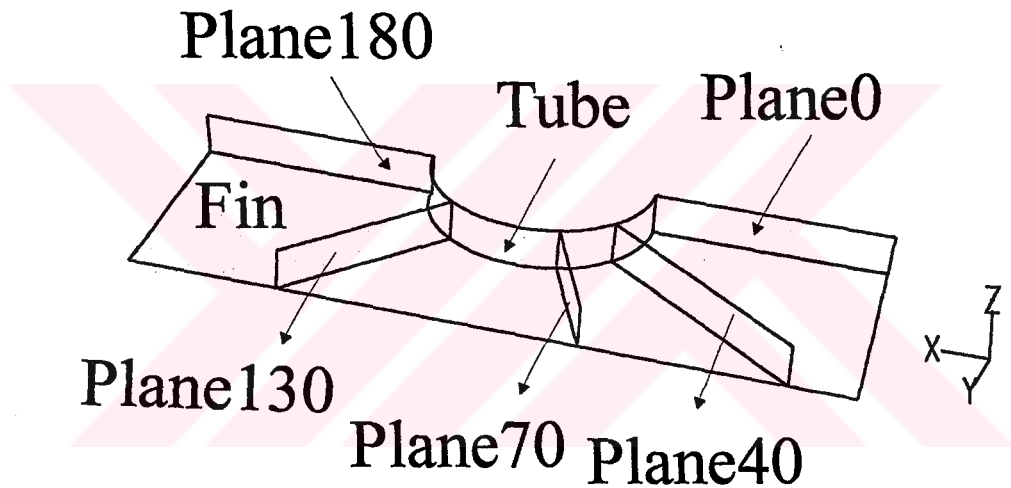
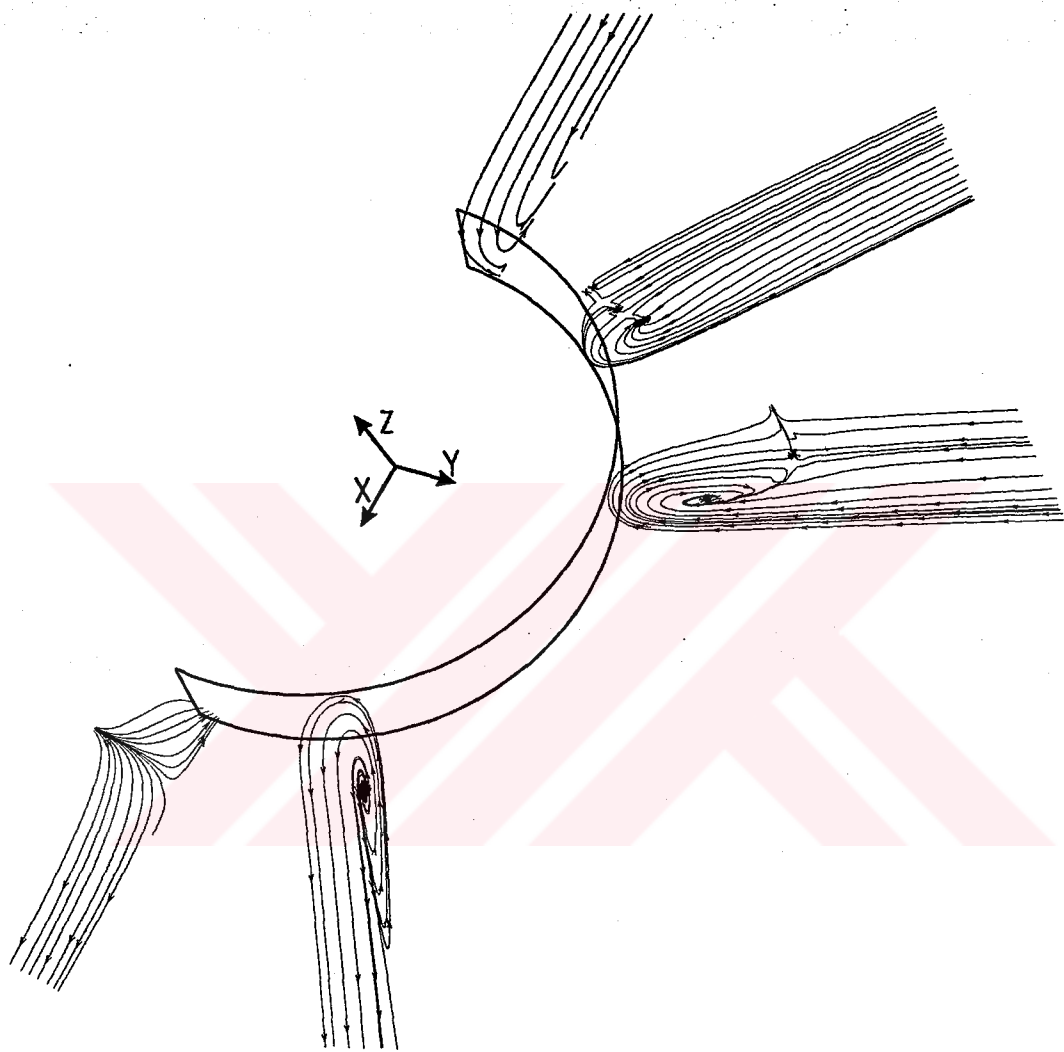


Figure 5.34. Location of theta-cut planes created around the tube surface

Figure 5.35 shows the horseshoe vortex evolution in the circumference of the tube for the single row case at various theta-cut planes at different time frames. In Fig. 5.35a streamlines are shown in the Plane0, Plane40, Plane80, Plane130, and Plane180 around the tube at $\tau = 2$. The flow is in the x -direction. As the flow approaches the tube, at Plane0, the streamlines hit the tube and focus around a point near the tube- fin junction. This focus is of separation type and represents the origin of the horseshoe vortex. At Plane40, there are 6 singular points close to the tube surface. The focus of separation seen in the Plane0 comes close to the tube surface and a nodal point of separation on the top of the focus appears. A saddle point

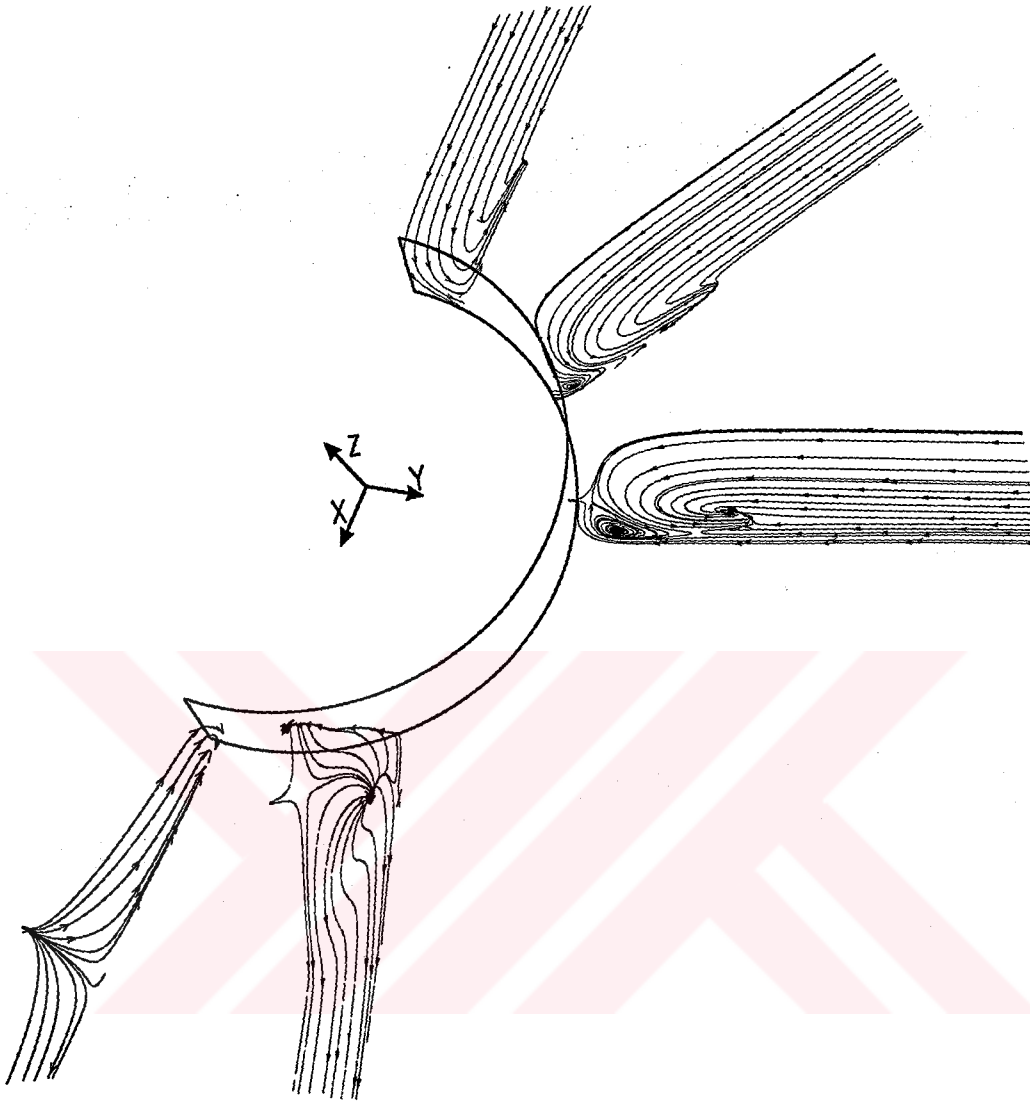
separates these two singular points. There are 2 nodal points of separation side by side and they are separated from the previous nodal point by a saddle point between them. At Plane80, the focus of separation with a clockwise sense of rotation becomes more evident with its saddle point. This focus is followed by a nodal point of separation and a line of separation each located on top of the saddle point. At the downstream part of the tube, at Plane130, streamlines emerging from a separation line adjacent to the fin surface encounter the tube wall and form a focus near the tube-fin junction in the form of a horseshoe type vortex as observed before in Fig 5.33a. From Fig. 5.35a, it is evident that the origin of the separation line is on the fin surface. At Plane180, a nodal point of attachment near the tube wall is also clearly seen. Figure 5.35b shows distribution of streamlines obtained at Plane0, Plane40, Plane60, Plane130, and Plane180 at $\tau = 6$. At $\tau = 6$, a stronger focus of separation at all theta-cut planes in the upstream part of the tube compared with those obtained at $\tau = 2$ can be clearly seen. At all theta-cut planes a corner vortex rotating in the clockwise sense is formed near the tube and the fin junction cause the center of the focus to shift a little away from the tube surface. The corner vortex propagates and becomes stronger towards the top of the tube. There are two saddle points each located to the right and to the left of the corner vortex. The one to the right prevents the corner vortex and the focus from interacting with each other. Similarly, the one to the left prevents the corner vortex from interacting with the main flow causing it to become attached to the tube wall. At Plane130, singular points interactions, such as a saddle point, a focus and two nodes, can be observed in that, lines emerging from a focus of attachment and a nodal point of attachment are feed into a nodal point of separation. At Plane180, streamlines emerging from a nodal point of attachment focus near the tube and the fin junction. At $\tau = 17$, the horseshoe vortex rotating in the counterclockwise direction formed in front of the tube surface with its saddle point can be very clearly seen at all theta cut planes (Fig. 5.35c). At Plane60, a corner vortex pushes the horseshoe vortex further down causing it to elongate in the flow direction. At the same time, a clockwise rotating vortex located behind the horseshoe vortex prevents the horseshoe vortex from displacing further away from the tube. At Plane80, both the corner vortex and the clockwise- rotating vortex are

elongated. As a result the horseshoe vortex becomes stronger and more compact (Tutar and Akkoca, 2004).



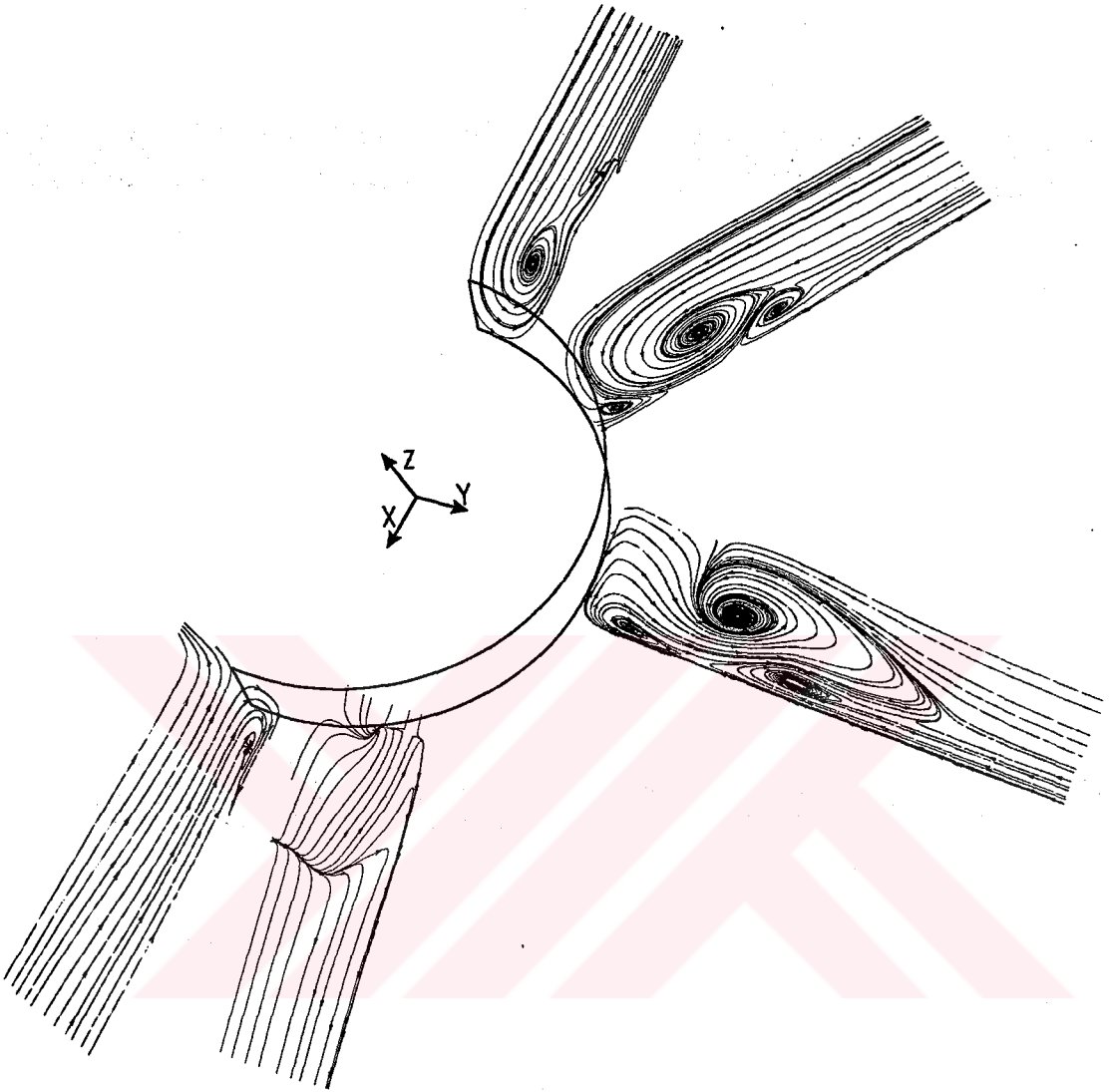
(a)

Figure 5.35a- Horseshoe vortex evolution in the circumference of the tube for the single row case at various theta-cut planes, $\tau=2$



(b)

Figure 5.35b- (continued) Horseshoe vortex evolution in the circumference of the tube for the single row case at various theta-cut planes, $\tau = 6$

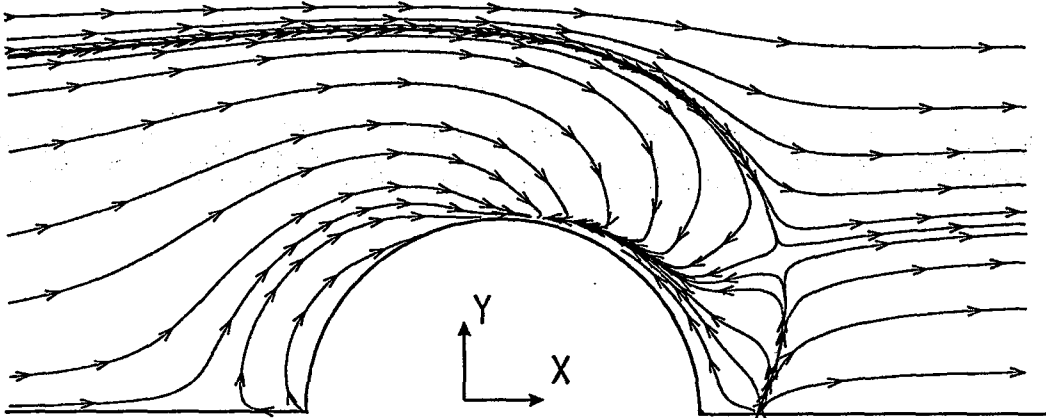


(c)

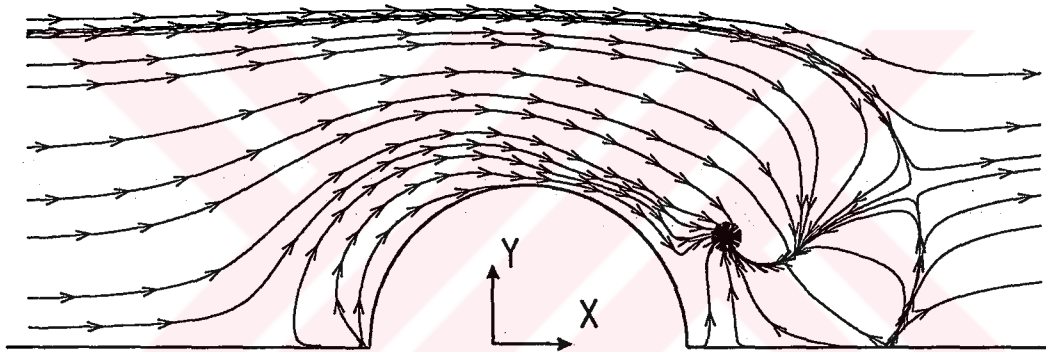
Figure 5.35c- (continued) Horseshoe vortex evolution in the circumference of the tube for the single row case at various theta-cut planes, $\tau = 17$

Figure 5.36 shows streamlines on the fin surface obtained at different time scales at a) $\tau = 2$, b) $\tau = 6$, c) $\tau = 17$, respectively. It can be seen from Fig. 5.36a that the streamlines emerging from a saddle point and a focus of attachment in the downstream of the tube on the fin surface form a separation line close to the tube surface which corresponds to the origin of the one observed previously in Fig. 5.35a at the theta130 plane. As the lines emerging from this separation line hit the tube wall, they form a horseshoe type vortex behind the tube as it is seen in Fig. 5.35a. In

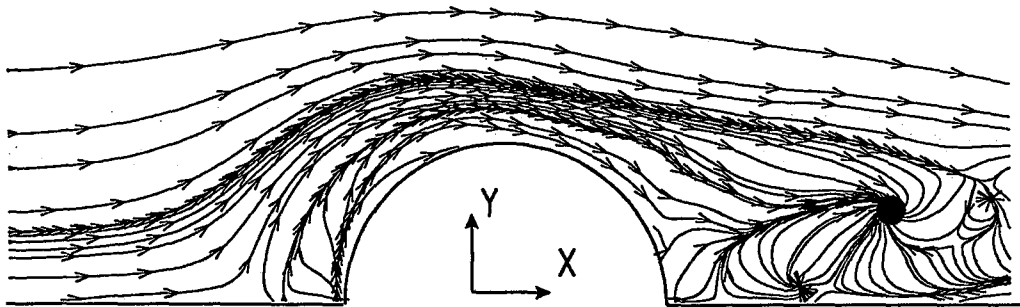
Fig. 5.36b, the saddle point shifts in the downstream direction from its previous position obtained at $\tau = 2$. Some of the lines emerging from the saddle point forms a separation line ahead of it. These lines feed into a focus of separation near the tube surface. The singular point interactions can also be seen at theta130 plane in Fig.5.35b at the same instant. At $\tau = 17$, there are strong singular point interactions on the fin surface in the wake of the tube as shown in Fig. 5.36c.



(a)



(b)



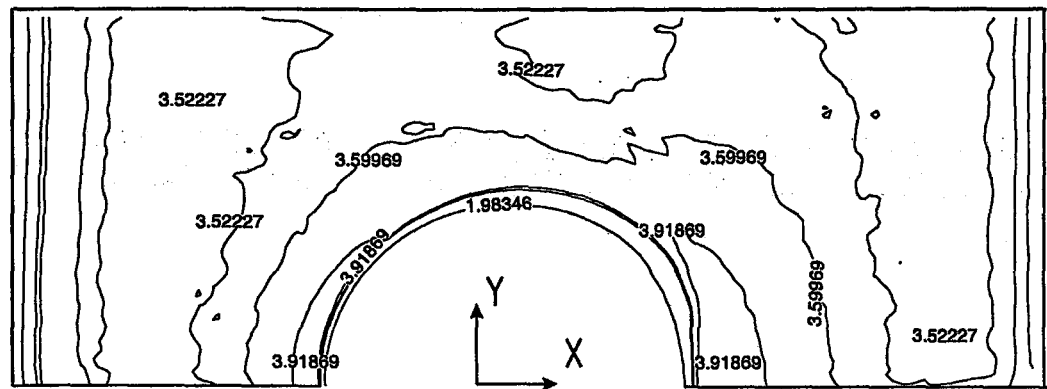
(c)

Figure 5.36. Streamlines on the fin surface for the single row case, a) $\tau = 2$; b) $\tau = 6$; c) $\tau = 17$

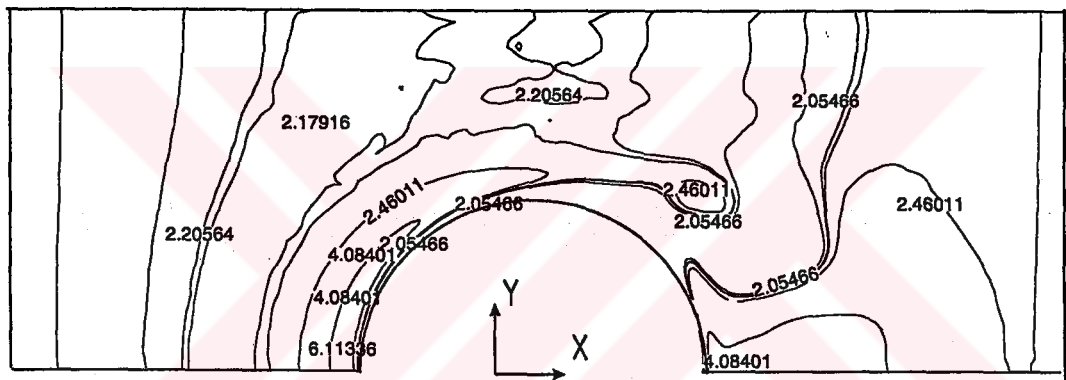
The heat transfer coefficients on the fin surface are obtained at $\tau = 2, 6$, and 17 as shown in Fig. 5.37a, b, c, respectively. At $\tau = 2$, (Fig. 5.37a) the heat transfer coefficients on the forward and backward parts of the tube are identical and have a value of $3.91869 \text{ W/m}^2\text{K}$ due to the existence of horseshoe vortices observed previously in Fig. 5.33a. The horseshoe type vortex observed in Fig. 5.35a at $\theta = 130^\circ$ causes the local heat transfer coefficient to increase in the corresponding region on the fin surface. That's why fin heat transfer coefficient in that region is obtained as large as in front of the tube and has a value of $3.91869 \text{ W/m}^2\text{K}$ (Fig. 5.37a). At $\theta = 180^\circ$, streamlines emerging from nodal point of attachment seem to focus around a point very close to the tube and the fin junction as seen in Fig. 5.35a. This focus is also the main cause of the same amount of heat transfer coefficient of $3.91869 \text{ W/m}^2\text{K}$ that obtained in front of the tube (Fig. 5.37a). At $\tau = 6$, (Fig. 5.37b), higher fin heat transfer coefficients are obtained due to the stronger and more compact horseshoe vortices and foci as observed previously in Fig. 5.35b and 5.33b. The propagating corner vortex towards the top of the tube observed previously in Fig. 5.35b cause a heat transfer decrease at about $2.05466 \text{ W/m}^2\text{K}$ as shown in Fig. 5.37b. The lower heat transfer coefficients encountered in the downstream part of the tube, on the fin surface, such as $2.05466 \text{ W/m}^2\text{K}$, can be attributable to the singular point interactions as shown in Fig. 5.36b. The focus at $\theta = 180^\circ$ plane shown in Fig. 5.35b is the main cause of heat transfer increase in the corresponding region on the fin surface, $4.08401 \text{ W/m}^2\text{K}$. The counterclockwise- rotating horseshoe vortex formed in front of the tube as shown in Fig. 5.35c cause an increase in the fin heat transfer coefficient in front of the tube. The magnitude of the local heat transfer coefficients at the θ -cut planes shown in Fig. 5.35c correspond to the corner vortex, the counterclockwise rotating horseshoe vortex and the clockwise rotating vortex are obtained as $1.05589 \text{ W/m}^2\text{K}$, $6.10671 \text{ W/m}^2\text{K}$, and $1.75206 \text{ W/m}^2\text{K}$ respectively (Fig. 5.37c). Therefore it is concluded that the extended horseshoe vortex is the main cause of higher heat transfer coefficients while vortices other than the horseshoe vortex are the main cause of lower heat transfer coefficients encountered in the upstream part of the tube. The corner and the clockwise rotating vortices consume the surrounding fluid's temperatures while they are growing. That's

s why fin heat transfer coefficients decrease during the horseshoe vortice's growth process. Poor heat transfer in the downstream part of the tube is attributed to a strong interactions between singular points (Fig. 5.36c) and the dead flow region.

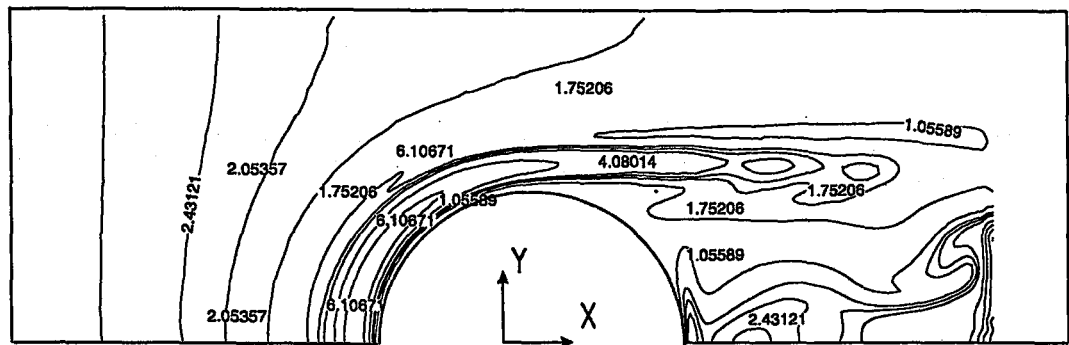




(a)



(b)



(c)

Figure 5.37. Local heat transfer coefficients on the fin surface for the single row case, a) $\tau = 2$; b) $\tau = 6$; c) $\tau = 17$

Figure 5.38 shows instantaneous vorticity contours around the tubes very close to the fin surface at $Z = 0.003$ plane for the staggered layout at non-dimensional time steps of $\tau = 8, 100$, and 400 . At $\tau = 8$, the local flow structure for each cylinder looks similar to that for the single row configuration. There are three main vortices observed: Firstly, a horseshoe vortex in the forward part of the tubes, secondly, a vortex in the backward part of the tubes, and thirdly, a vortex in the wake of the tubes near the tube- fin junction. The maximum vorticity is obtained as $3.25 \times 10^4 \text{ s}^{-1}$ at $\tau = 8$. At $\tau = 100$, vortices are stacked in front of each tube. The maximum vorticity has a value of $8.85 \times 10^4 \text{ s}^{-1}$. All of the horseshoe vortices are stretched towards the downstream direction causing an increase in the heat transfer coefficients in their effective area. The strongest horseshoe vortices are observed in front of the second row. At $\tau = 100$, it is found to be a large amount of ineffective area occurs especially in the downstream parts of each tube. At $\tau = 400$, the horseshoe vortices are elongated more in the flow direction. Ineffective areas are disappeared and a number of vortices are observed especially around the fourth row washing away into the wake. The maximum value of the vorticity is found to be $1.13 \times 10^5 \text{ s}^{-1}$.



Figure 5.38. Patterns of vorticity topology for the staggered arrangement of the 4 cylinder configuration at $Z = 0.003$ plane at different non-dimensional times: a) $\tau = 8$; b) $\tau = 100$; c) $\tau = 400$

Figure 5.39 shows local heat transfer coefficients on the fin surface. At $\tau = 8$ there is no significant difference in terms of heat transfer coefficients obtained in the forward and the backward parts as well as at the apex of the cylinders on the fin surface, since the vortex formed in the backward part of the cylinder is in the horseshoe form and contributes to increase in the fin heat transfer coefficient (Fig.5.39a). At $\tau = 100$ the highest fin heat transfer coefficient is obtained in front of the second cylinder as shown in Fig.5.39b which has a value of $74.3619 \text{ W/m}^2\text{K}$ due to the strongest horseshoe vortices obtained in front of the second cylinder as shown in Fig. 5.38b. From Fig. 5.38b, ineffective area occurs especially in the downstream parts of each cylinder can be clearly seen. Therefore the fin heat transfer coefficients are obtained as low as $7.92739 \text{ W/m}^2\text{K}$ (Fig.5.39b) in the corresponding ineffective area regions. At $\tau = 400$ increased heat transfer coefficients are obtained as compared to the previous time ($\tau = 100$) because of the elongated horseshoe vortices as observed in Fig. 5.38c (Fig. 5.39c). The heat transfer coefficients in front of the second and the fourth cylinders are found to be the same which are $74.3619 \text{ W/m}^2\text{K}$.

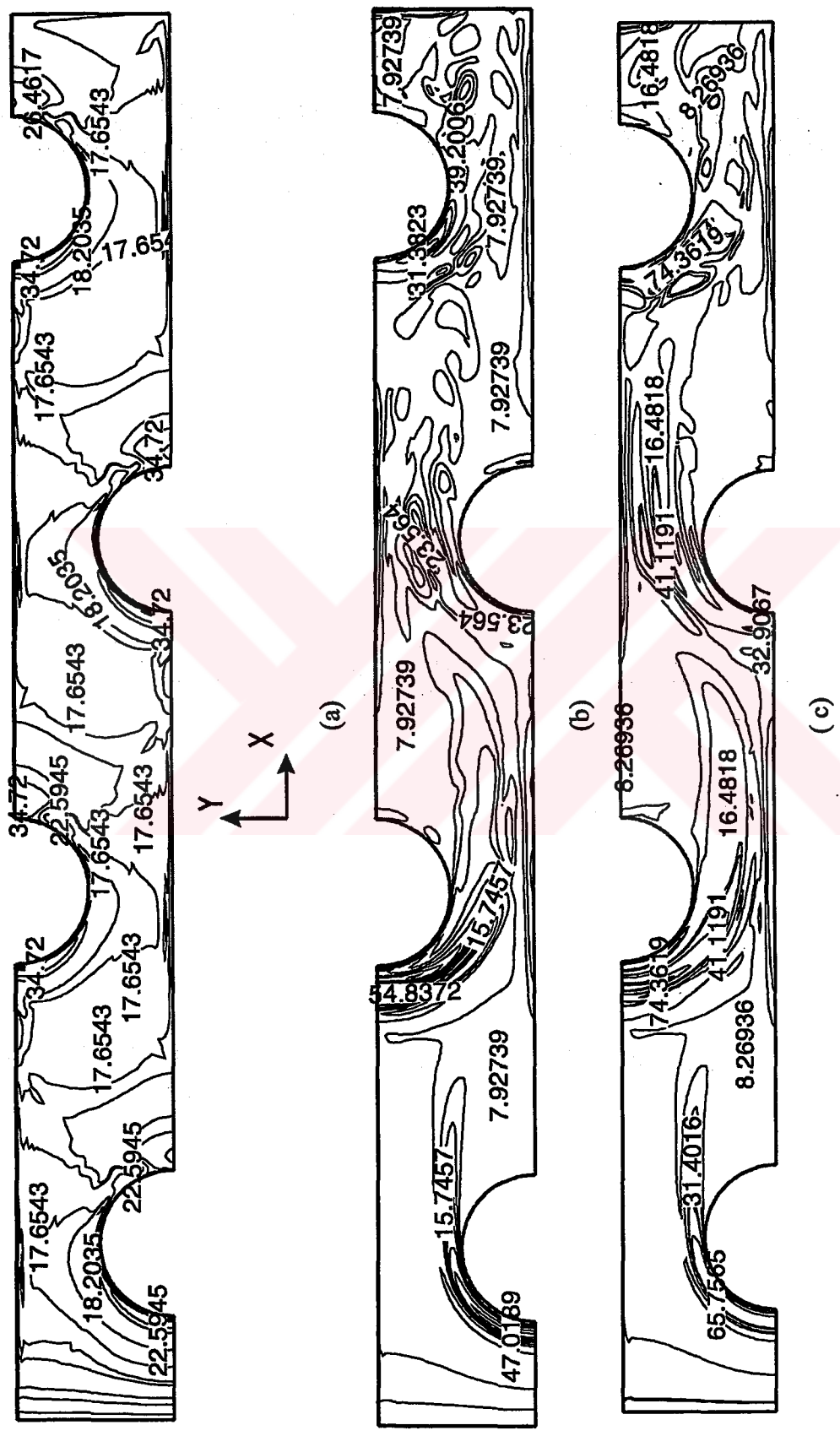


Figure 5.39. Contours of heat transfer coefficients on the fin surface for the 4 cylinder, staggered layout at: a) $\tau = 8$ b) $\tau = 100$; c) $\tau = 400$

Figure 5.40 illustrates the horseshoe vortex evolution in the circumference of the second cylinder at various theta-cut planes at $\tau = 100$ and 400. The flow structure around the second cylinder is rather complex because it is affected by the presence of the first and the third cylinders. At Plane0, two horseshoe vortices are observed: One is near the forward stagnation point of the tube and the other is behind the first one. There are also two saddle points between the horseshoe vortices although they are not appear clearly. At theta50, two horseshoe vortices are separated by a saddle point located between them. The horseshoe vortices are elongated and move in a direction downstream of the second cylinder contributing high heat transfer coefficients around the cylinder as shown in Fig. 5.38b and Fig. 5.39b and discussed before. Following these horseshoe vortices, two counter-rotating vortices can also be seen. Because of the saddle point located in between a pair of vortices near the cylinder and the counter-rotating vortices, these combination of vortices remain a part. A saddle point try to keep this pair of vortices in the vicinity of the cylinder wall. At Plane150, behind the tube, there is a vortex with clockwise sense of rotation near the tube wall. At Plane180, there is also a horseshoe type vortex and a focus of separation. A saddle point between them prevents the horseshoe vortex and the focus from interacting with each other. The evolution of the horseshoe vortex around the second cylinder at $\tau = 400$ is shown in Fig. 5.40b: A focus of separation near the cylinder and the fin junction is followed by a horseshoe vortex at Plane0. There are two small vortices both rotating in the counterclockwise direction behind this horseshoe vortex. At Plane50, the focus located near the junction changes its location in a direction away from the cylinder allowing the horseshoe vortex to come close to the cylinder wall thus causing an increase in the heat transfer coefficient in that region. A pair of counter-rotating vortices and a small focus near the fin surface as well as a saddle point in the backward part of the horseshoe vortex can also be seen. At Plane180, a horseshoe type vortex with its saddle point and two foci of separation adjacent to the fin surface can be also clearly seen. It is important to say here that, there is a very close interaction between the local flow characteristics and the heat transfer data as the flow characteristics and the heat transfer data are compared with each other at different instant of times such as: The heat transfer coefficient is

increased from $54.8372 \text{ W/m}^2\text{K}$ to $74.3619 \text{ W/m}^2\text{K}$ in the forward part of the second cylinder. Similarly, heat transfer coefficient towards the apex of the second cylinder is increased from $15.745 \text{ W/m}^2\text{K}$ to $41.1191 \text{ W/m}^2\text{K}$. The values are also increased in the backward part of the cylinder from $7.92739 \text{ W/m}^2\text{K}$ to $8.26936 \text{ W/m}^2\text{K}$. This amount of increments in the local heat transfer coefficients are strongly related to the flow behavior in the corresponding locations around the second cylinder (Tutar and Akkoca, 2004).

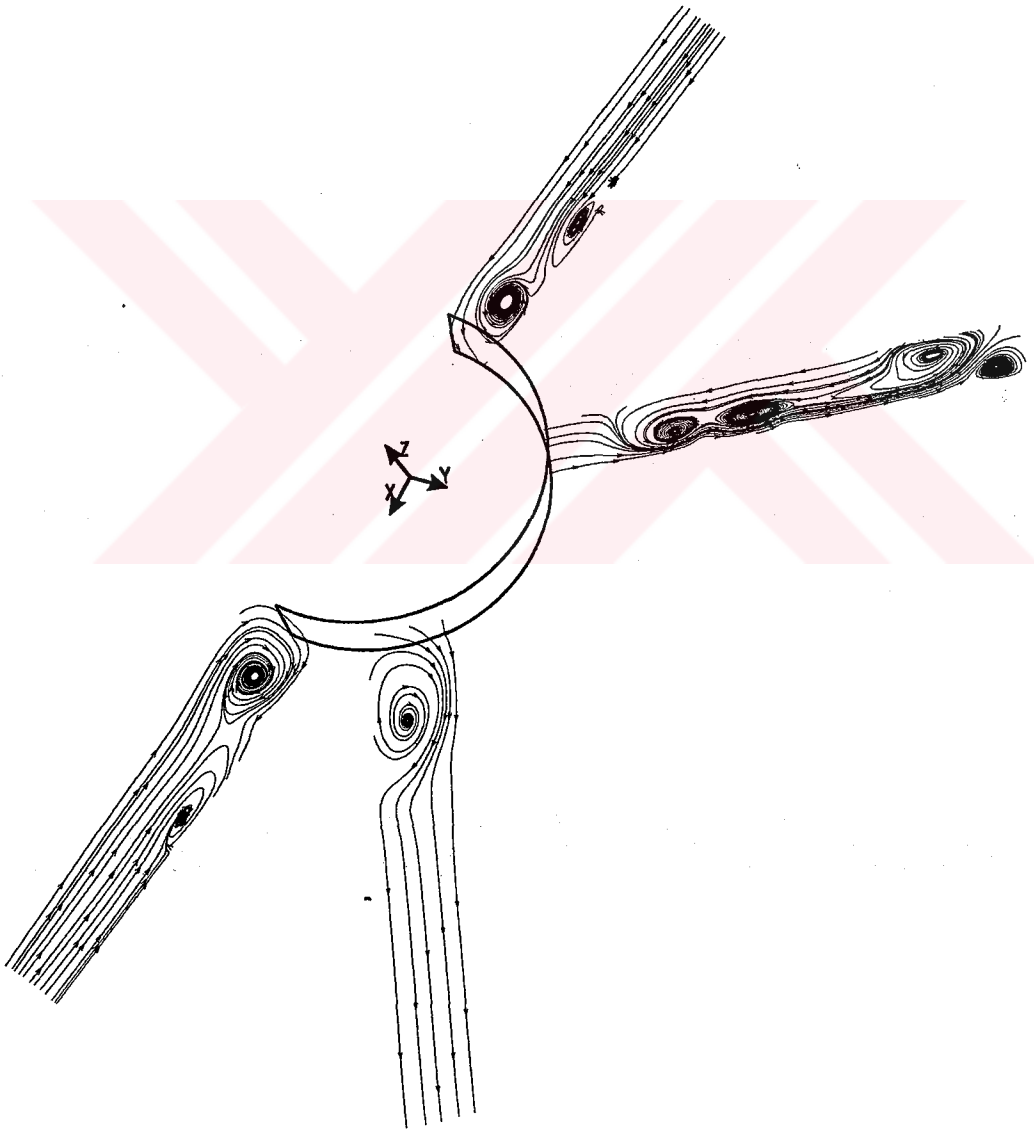


Figure 5.40a- Horseshoe vortex evolution in the circumference of the second cylinder for the staggered layout at various theta-cut planes, $\tau = 100$

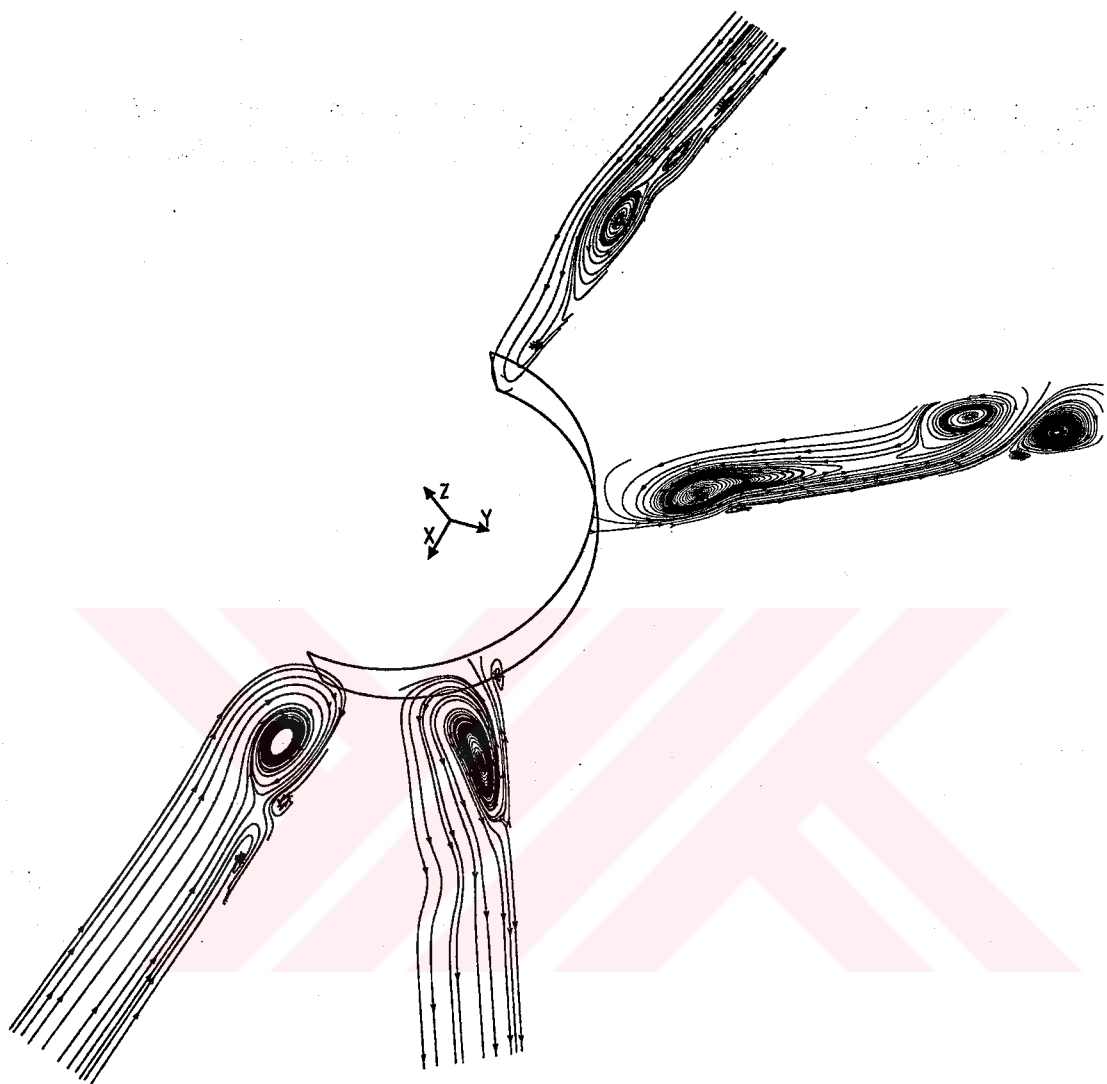


Figure 5.40b- Horseshoe vortex evolution in the circumference of the second cylinder for the staggered arrangement at various theta-cut planes, $\tau = 400$

5.7. Experiment Results

Figure 5.41 a, b, and c show the time averaged velocity vector, vorticity plot and streamline for the single cylinder configuration for $F_s = 0.365$ at $Re = 1460$ obtained from PIV experiment. The data illustrate side views and are taken at a plane close to the fin surfaces. The corresponding numerical results for the vorticity plots are shown in Fig. 5.41d. The position of the cylinder is indicated with black color as shown in Figures 5.41- 5.71. As the velocity vectors approaches the tube surface, they tend to turn back and focus in the vicinity of the tube wall, resulting in the formation of a horseshoe vortex. A saddle point lets the horseshoe vortex to appear in close vicinity of the tube wall and is visible in both experimental and numerical results. However, a focus of separation near the saddle point that is located on the fin surface is clearly seen in the numerical result while it is invisible in the experimental results. This is due to the fact that the image of the particles in the PIV experiment is only possible to be recorded near the wall but not exactly at the wall, that is, the focus of separation is out of field of view of the PIV measurement. However, it is possible to get the vorticity and the streamline data at the wall in the numerical studies. That's why the focus of separation is clear in the numerical result. As it is explained in Section 4.9, the heat exchanger model developed for the present numerical study is symmetric about the top symmetry plane as shown in Fig. 4.3. That's why only one horseshoe vortex is visible in the numerical study, while two horseshoe vortices are visible in the experimental study. It is evident that there is a strong agreement between the experimental and the numerical results.

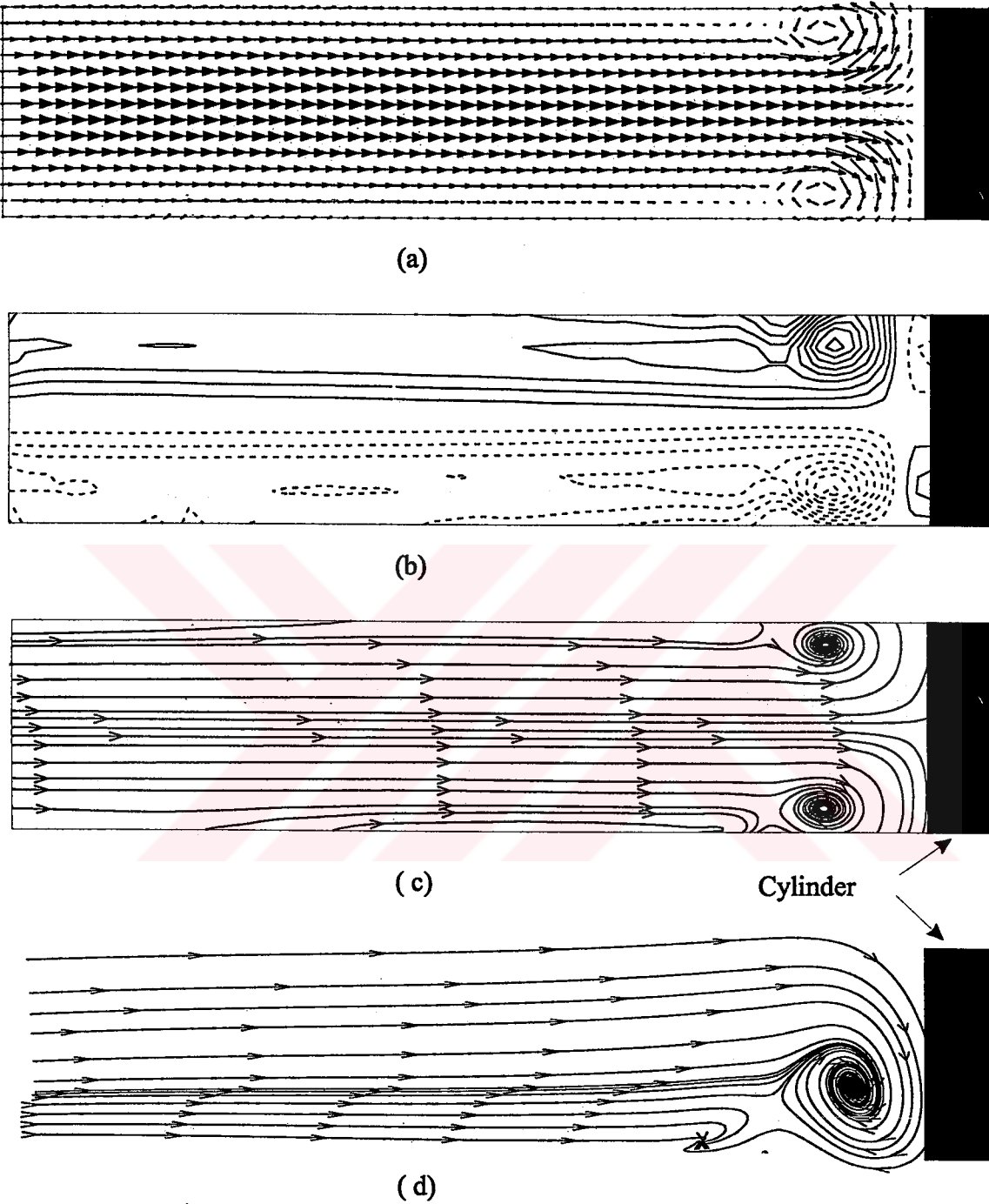


Figure 5.41. Comparison of experimental and numerical results for $Fs = 0.365$ at $Re = 1460$

- a) Time averaged velocity vector field (experimental result, side view)
- b) Time averaged vorticity contours (experimental result, side view)
- c) Time averaged streamline pattern (experimental result, side view)
- d) Time averaged streamline pattern (numerical study)

Figures 5.42- 5.47 illustrate instantaneous velocity vector field, streamline pattern, and vorticity contours for $F_s = 0.365$, at $Re = 600, 1460$ and 4000 at the centerline axis of the flow passage (side view) at frame numbers of $N = 6$ and $N = 8$. The positive and the negative vorticity is illustrated by solid and dashed lines, respectively. The locations of the horseshoe vortex near the vicinity of the tube surface is evident for each case. It is clear that as the Reynolds number is increased, stronger horseshoe vortices are observed near the tube wall. The second horseshoe vortex is clearly seen at $Re = 1460$ and $N = 6$ (Fig. 5.44). The negative and the positive vorticities are symmetric with each other.

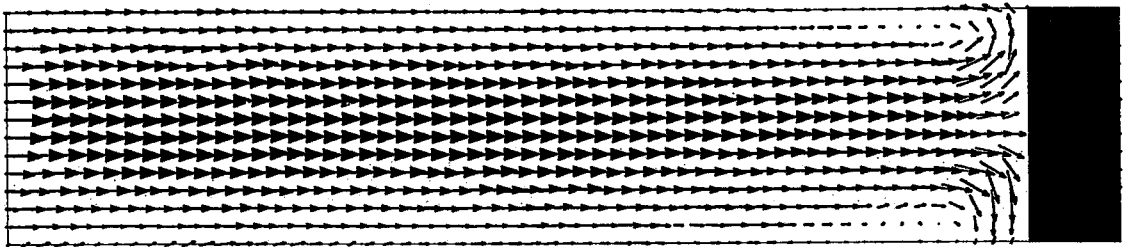
Figures 5.48, 5.49, and 5.50 shows the time averaged velocity vector field, streamline pattern and vorticity contours at the centerline of the flow passage for $F_s = 0.365$, at $Re = 600, 1460$, and 4000 , respectively (plan view). It is evident that the size of the wake region differs depending up on the Reynolds number. As the Reynolds number is increased, the size of the wake region becomes smaller for $F_s = 0.365$. The negative and the positive vorticities are symmetrically separated from the cylinder surface. More elongated vorticities are obtained for smaller Reynolds numbers. As the Reynolds number is increased, the vorticity contours become shorter but wider (Fig.5.50). The horseshoe vortices located at both sides of the cylinder are clearly seen at $Re = 1460$ (Fig. 5.49).

The instantaneous velocity vector, streamline, and vorticity contours at the centerline of the flow passage for $F_s = 0.365$ at different Reynolds numbers and frame numbers are presented in Figures 5.51-5.56. From these instantaneous data, strong singular point interactions are observed in the wake region at smaller Reynolds numbers. As the Reynolds number is increased, the formation of singular points decreases in the wake region for $F_s = 0.365$. At $Re = 4000$, the patterns of instantaneous vorticity differ from those obtained for $Re = 600$ and $Re = 1460$. The horseshoe vortices are clearly seen at $Re = 4000$ (Fig. 5.56).

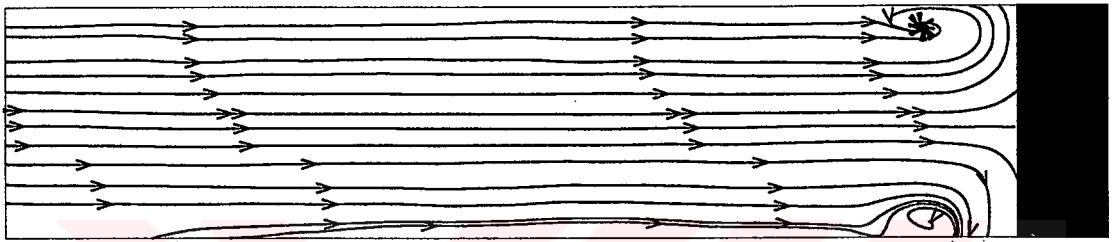
In Figures 5.57, 5.58, and 5.59, the time averaged flow data obtained on a plane close to the fin surface for $F_s = 0.365$ are given. The length of the wake region becomes smaller close to the fin surface when compared with the data obtained at the centerline of the flow passage.

Figures 5.60- 5.65 present instantaneous data obtained for $F_s = 0.365$ at different Reynolds numbers near the fin surface. As the flow characteristics obtained in the middle of the flow passage and near the fin surface are compared with each other, more singular point interactions can be observed near the fin surface. This is because of the effect of the fin surface strongly influences the flow behaviour in the flow passage. The vorticity contours are more elongated and the centers of the vorticities are closely spaced at $Re = 1460$ comparing to those of at $Re = 600$ (Figs. 5.61 and 5.62).

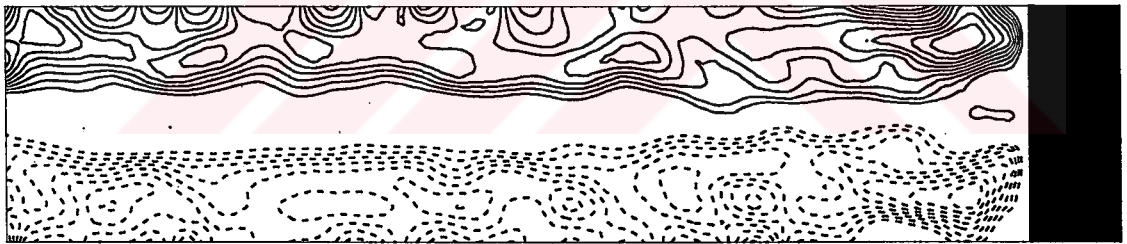
The time averaged data for $F_s = 0.116$ are presented in Figures 5.66 and 5.67. The size of the wake region and the formation of singular points decreases with an increase in the Reynolds number for small fin spacings. A number of singular points and a line of separation are obtained in the wake region for small fin spacings comparing to the larger fin spacings (Figs. 5.68 and 5.69). It is observed that the positive vorticity layer is longer while negative vorticity layer is shorter for small fin spacings (Figs. 5.70 and 5.71).



(a)

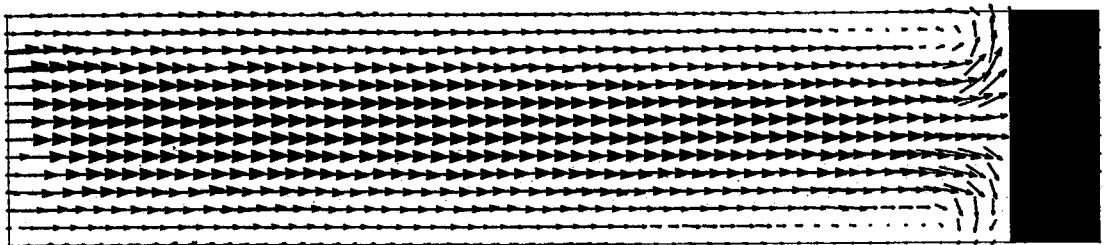


(b)

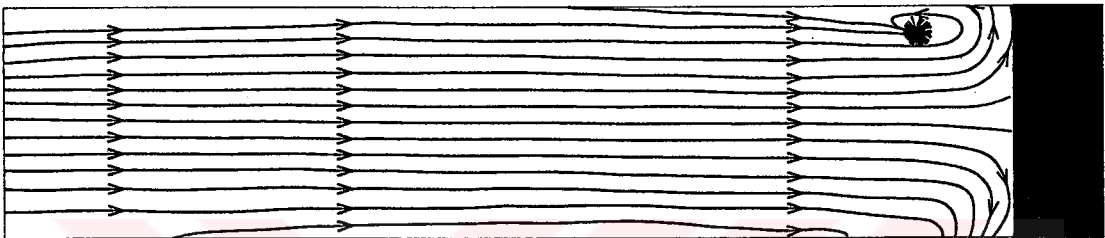


(c)

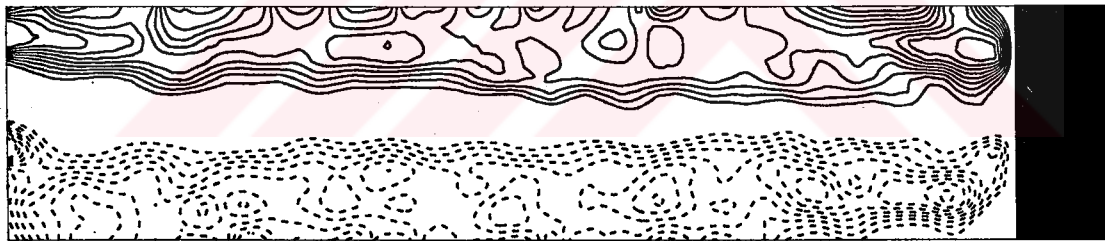
Figure 5.42. Experimental result for $Fs = 0.365$ at $Re = 600$, at $N = 6$, side view
(a) instantaneous velocity vector field
(b) instantaneous streamline pattern
(c) instantaneous vorticity contours



(a)

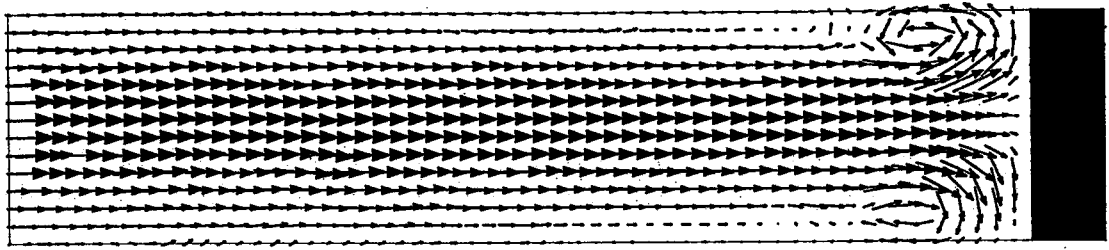


(b)



(c)

Figure 5.43. Experimental result for $Fs = 0.365$ at $Re = 600$, at $N = 8$, side view
(a) instantaneous velocity vector field
(b) instantaneous streamline pattern
(c) instantaneous vorticity contours



(a)

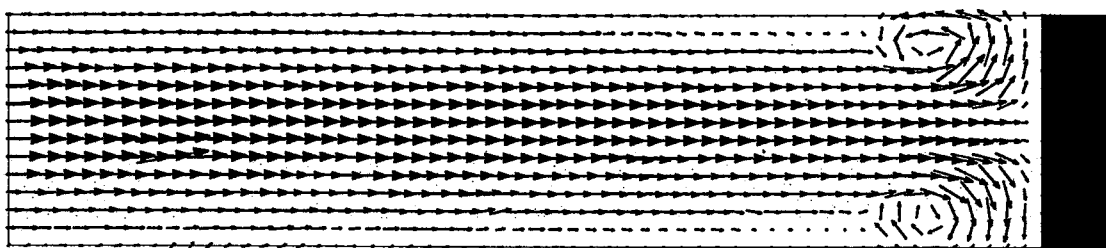


(b)

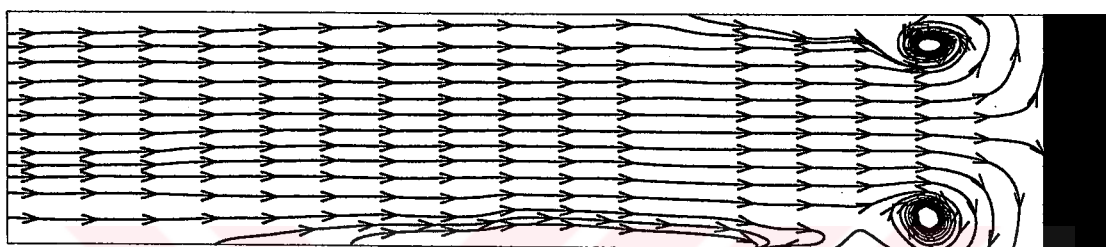


(c)

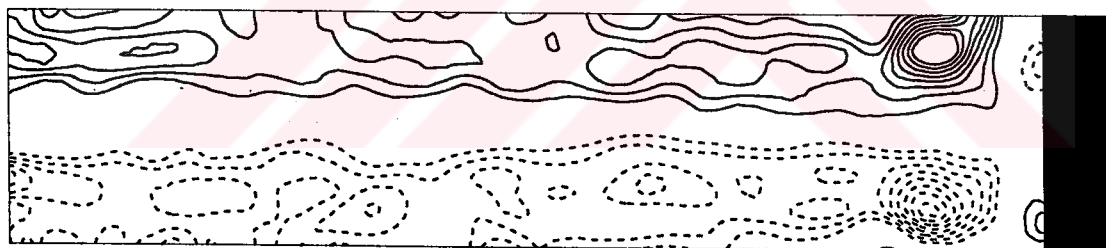
Figure 5.44. Experimental result for $Fs = 0.365$ at $Re = 1460$, at $N = 6$, side view
(a) instantaneous velocity vector field
(b) instantaneous streamline pattern
(c) instantaneous vorticity contours



(a)

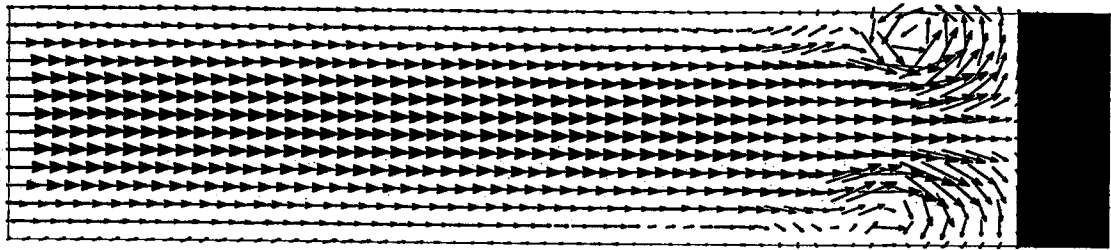


(b)

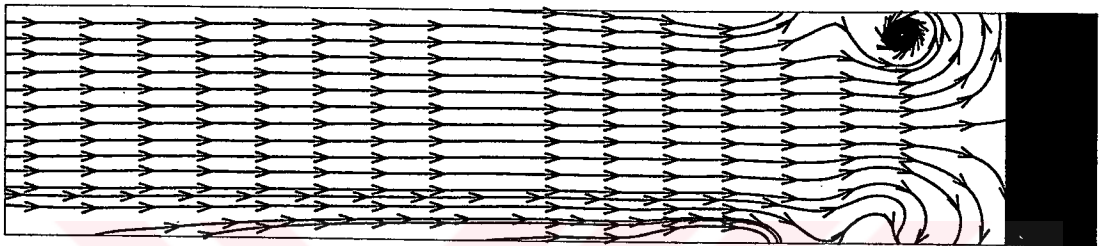


(c)

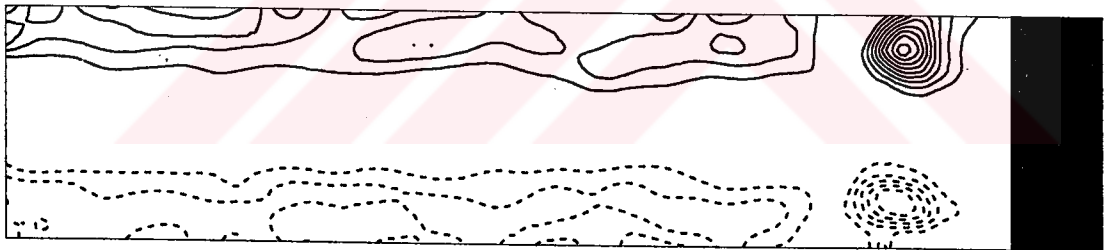
Figure 5.45. Experimental result for $Fs = 0.365$ at $Re = 1460$, at $N = 8$, side view
(a) instantaneous velocity vector field
(b) instantaneous streamline pattern
(c) instantaneous vorticity contours



(a)

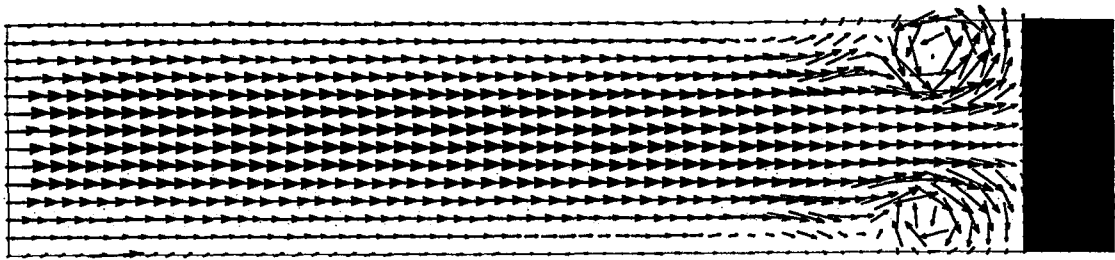


(b)



(c)

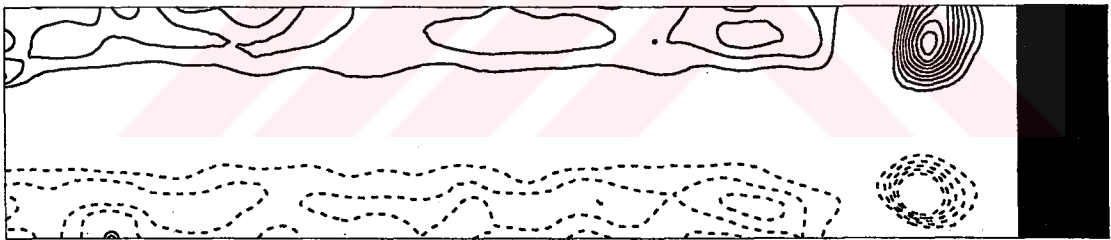
Figure 5.46. Experimental result for $F_s = 0.365$ at $Re = 4000$, at $N = 6$, side view
(a) instantaneous velocity vector field
(b) instantaneous streamline pattern
(c) instantaneous vorticity contours



(a)



(b)



(c)

Figure 5.47. Experimental result for $Fs = 0.365$ at $Re = 4000$, at $N = 8$, side view
(a) instantaneous velocity vector field
(b) instantaneous streamline pattern
(c) instantaneous vorticity contours

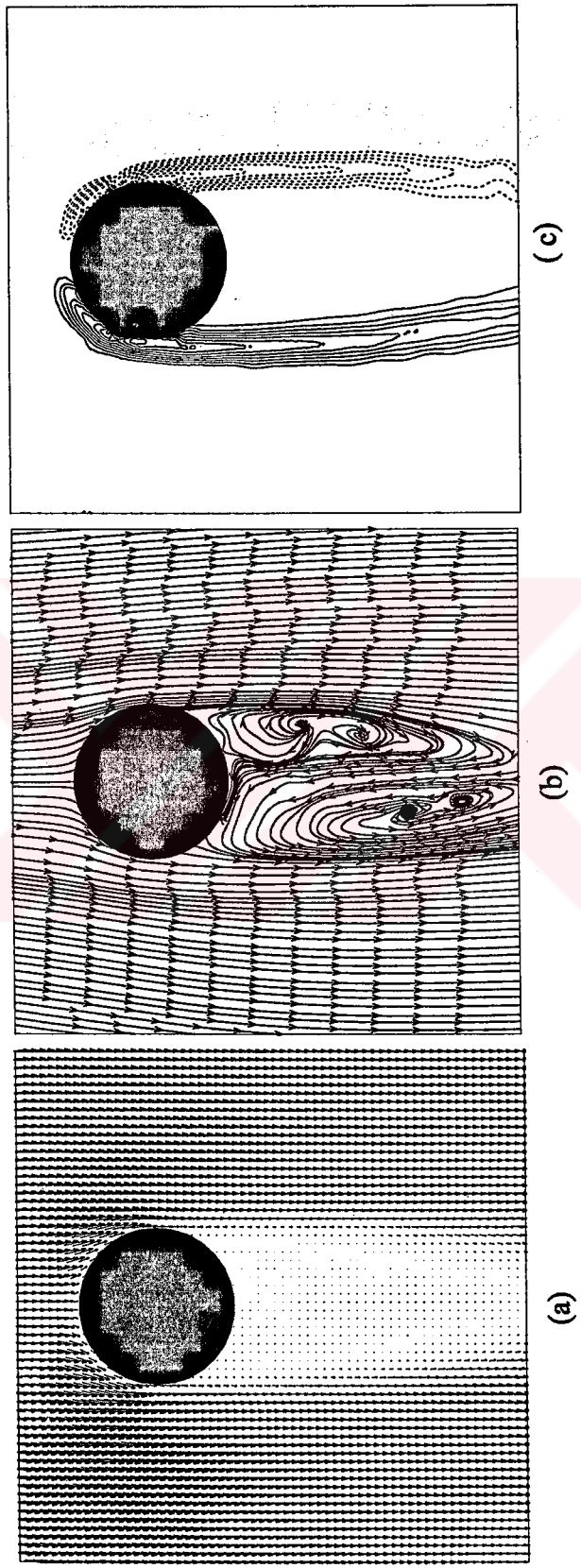


Figure 5.48. Experimental results obtained at the centerline of the flow passage for $F_s=0.365$ at $Re= 600$, plan view
(a) Time averaged velocity vector field
(b) Time averaged streamline pattern
(c) Time averaged vorticity contours

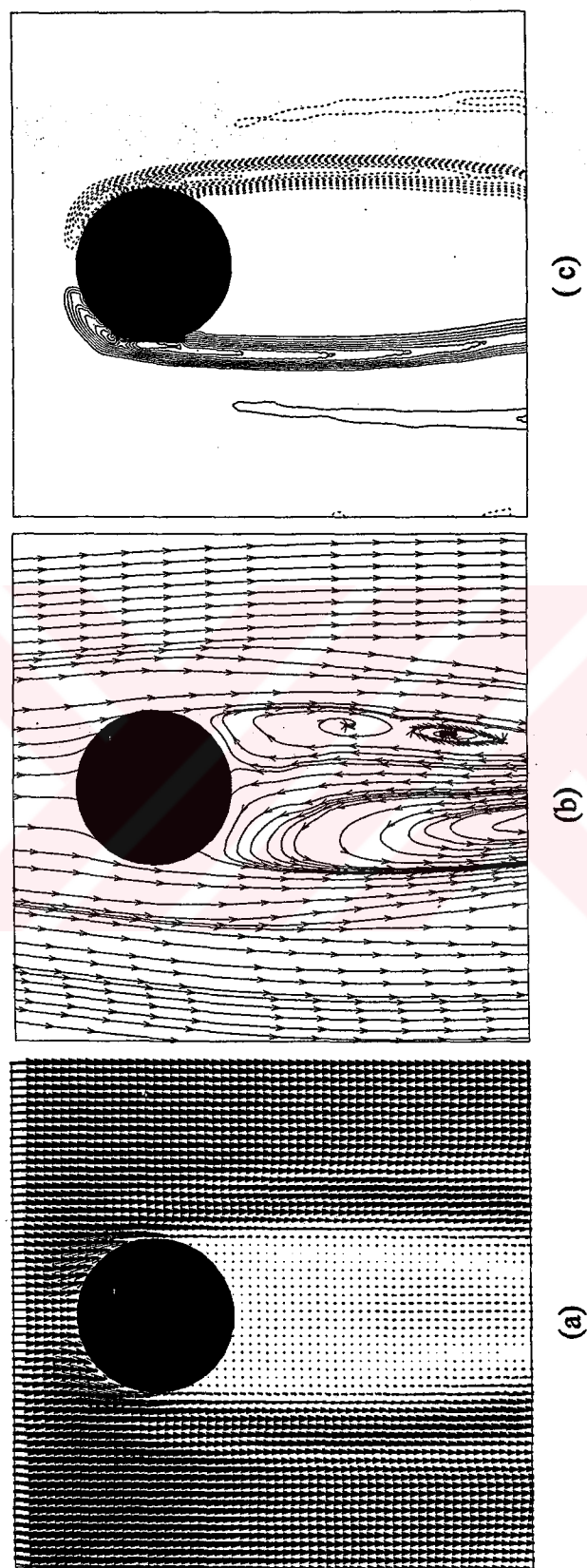


Figure 5.49. Experimental results obtained at the centerline of the flow passage for $Fr = 0.365$ at $Re = 1460$, plan view

- (a) Time averaged velocity vector field
- (b) Time averaged streamline pattern
- (c) Time averaged vorticity contours

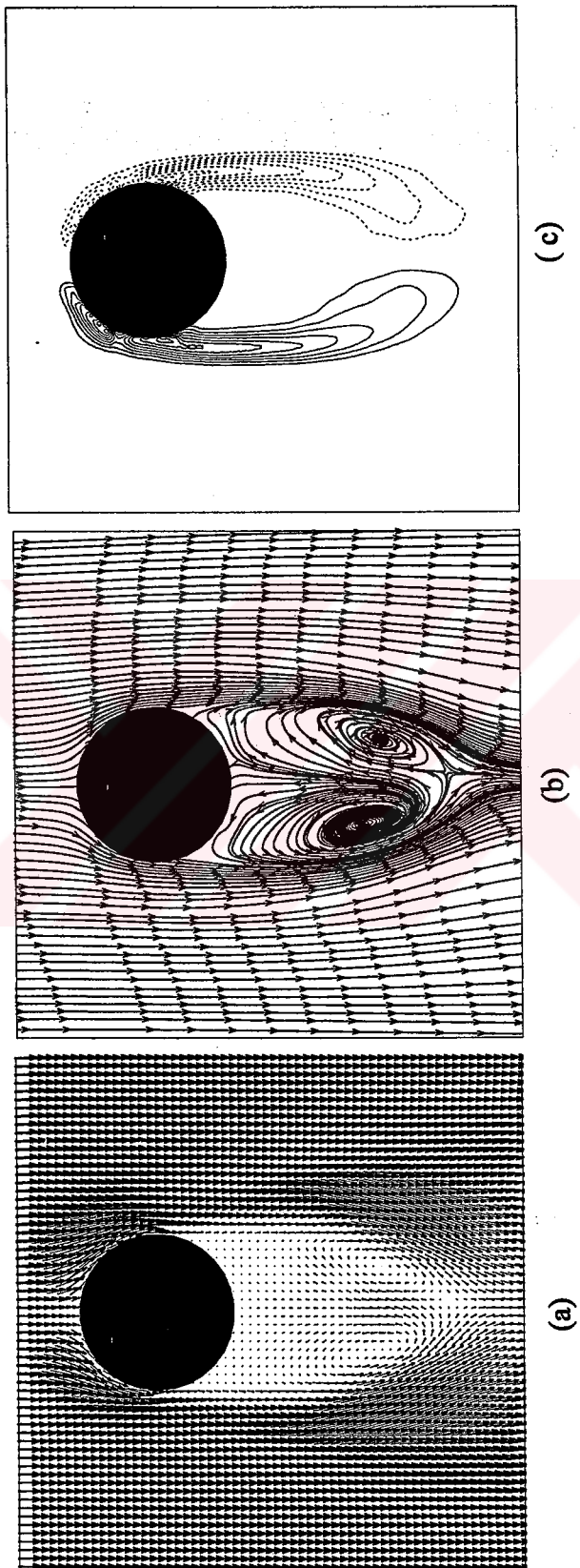


Figure 5.50. Experimental results obtained at the centerline of the flow passage for $Fs=0.365$ at $Re=4000$, plan view
(a) Time averaged velocity vector field
(b) Time averaged streamline pattern
(c) Time averaged vorticity contours

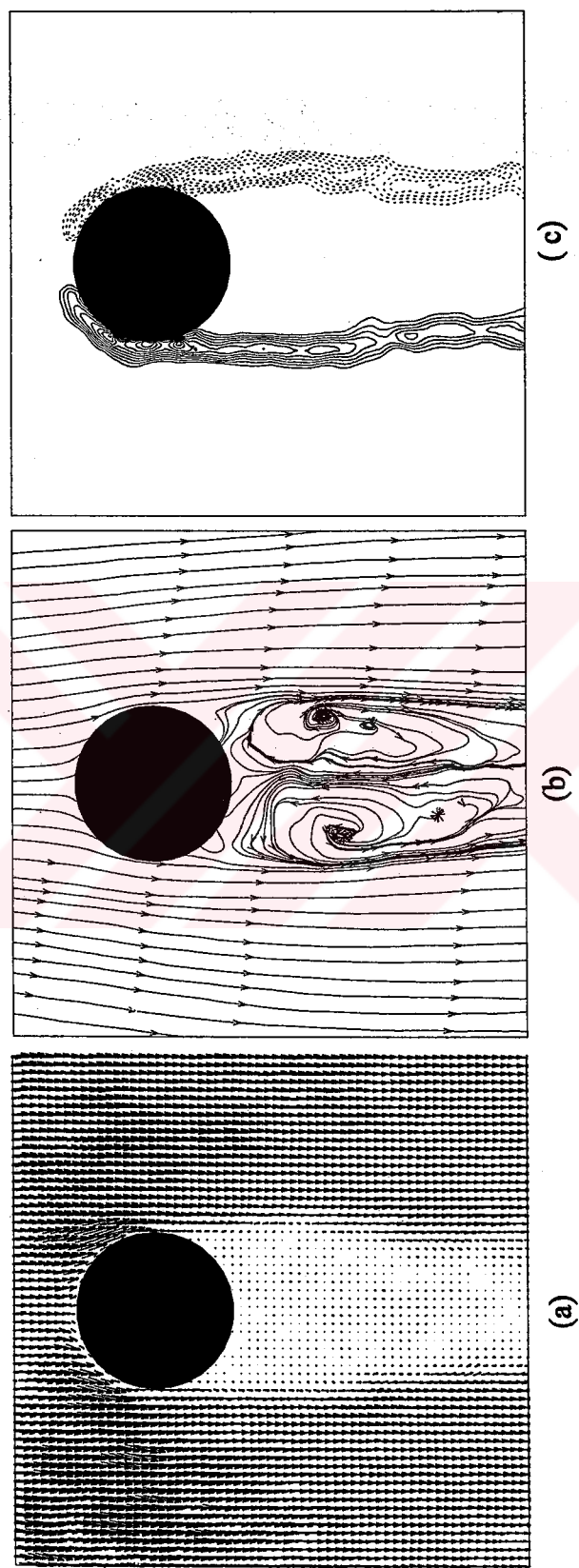


Figure 5.51. Experimental results obtained at the centerline of the flow passage for $Fs = 0.365$ at $Re = 600$ at $N = 6$, plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

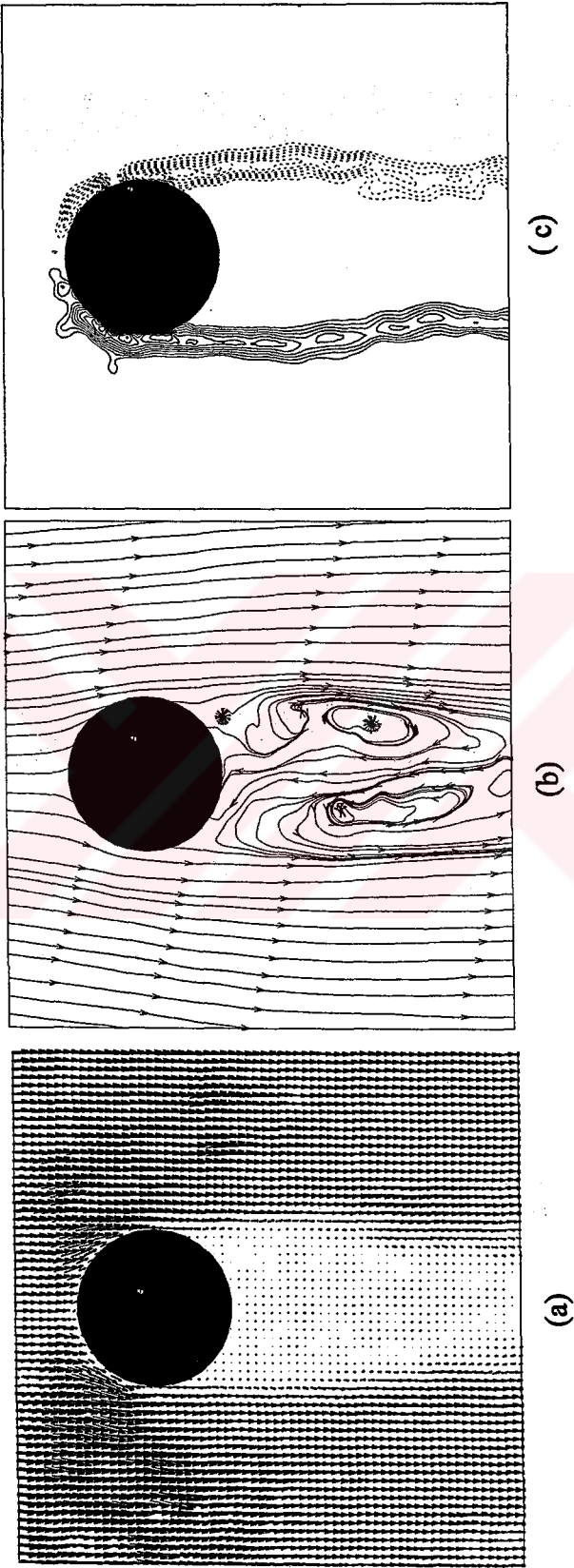


Figure 5.52. Experimental results obtained at the centerline of the flow passage for $F_s=0.365$ at $Re=600$ at $N=8$, plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

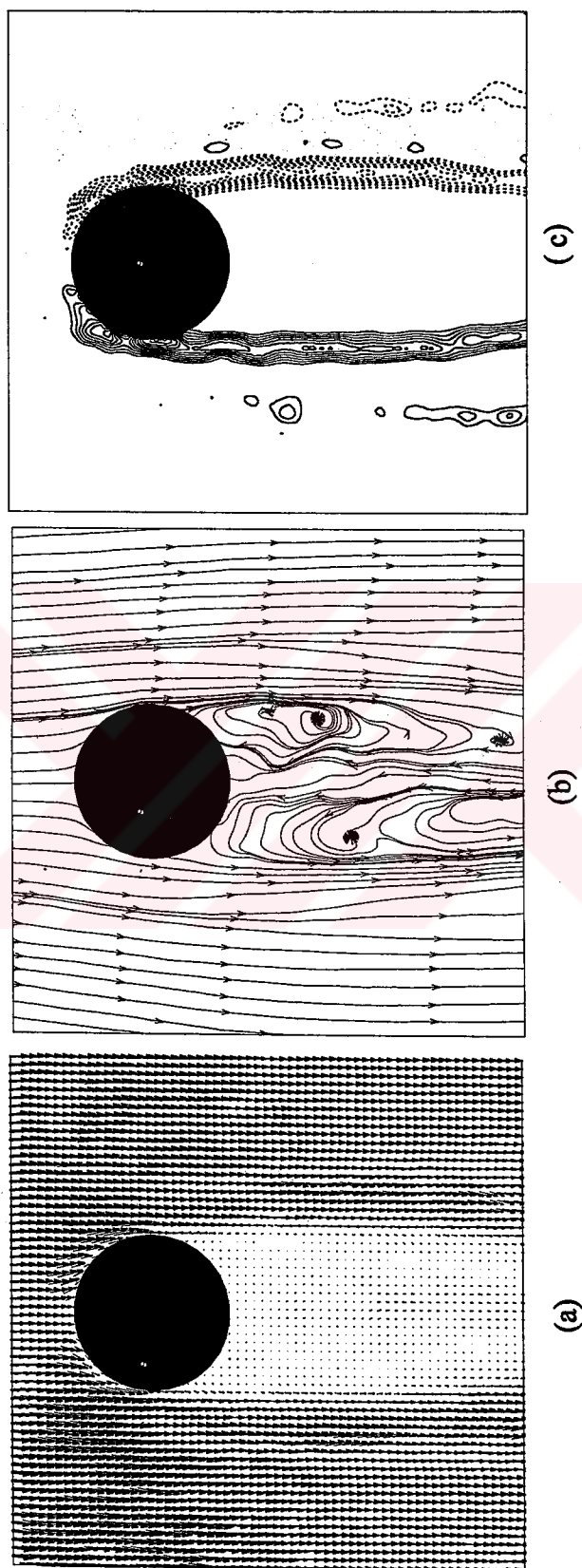


Figure 5.53. Experimental results obtained at the centerline of the flow passage for $F_s=0.365$ at $Re=1460$ at $N=6$, plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

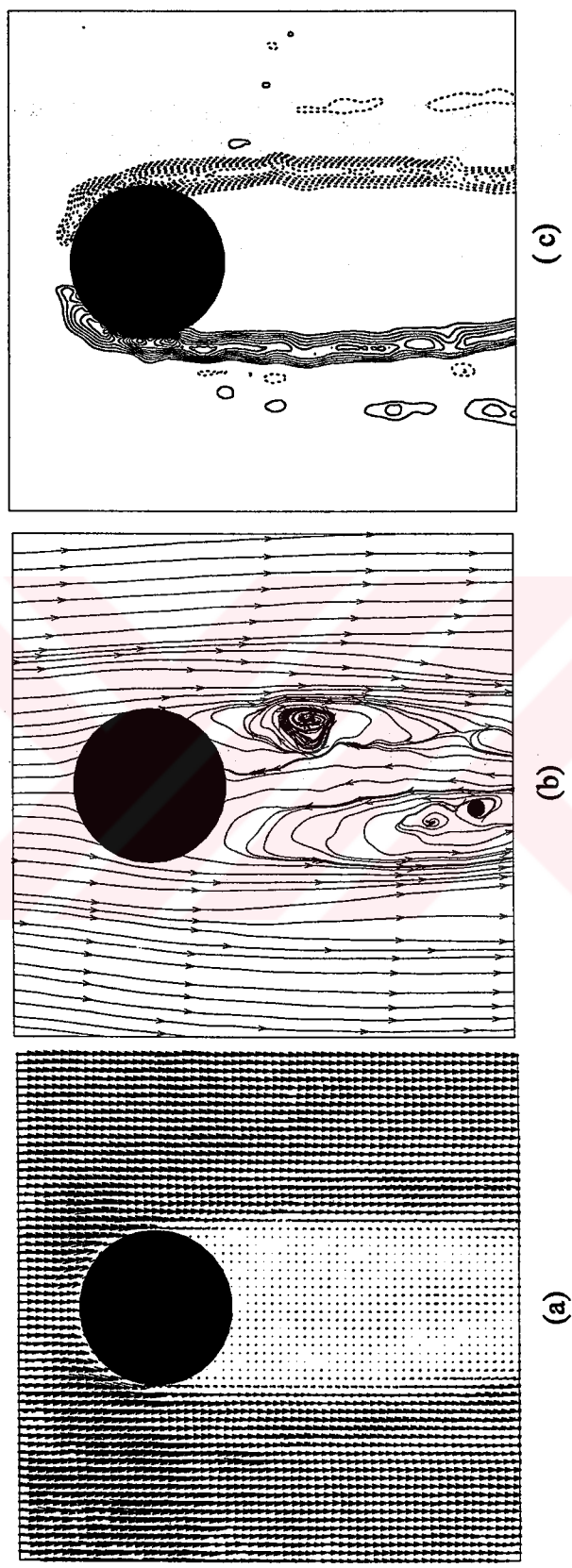


Figure 5.54. Experimental results obtained at the centerline of the flow passage for $Fs=0.365$ at $Re=1460$ at $N=8$, plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

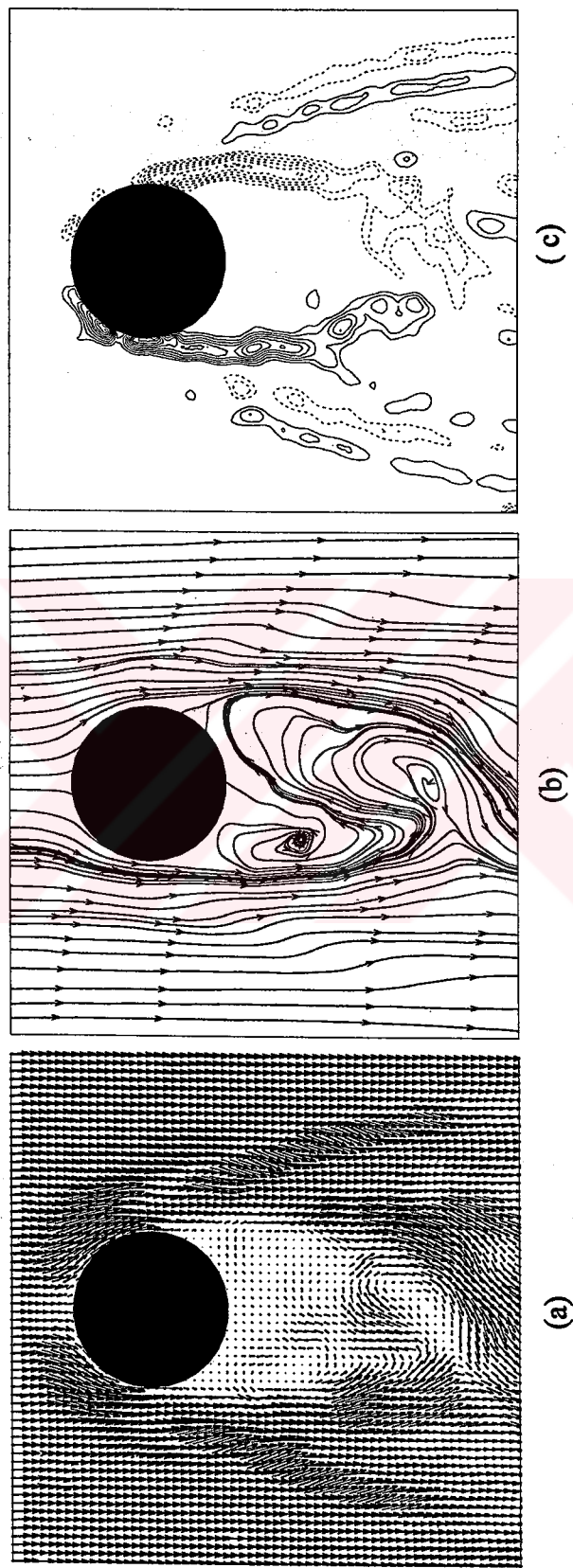


Figure 5.55. Experimental results obtained at the centerline of the flow passage for $F_s=0.365$ at $Re=4000$ at $N=6$, plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

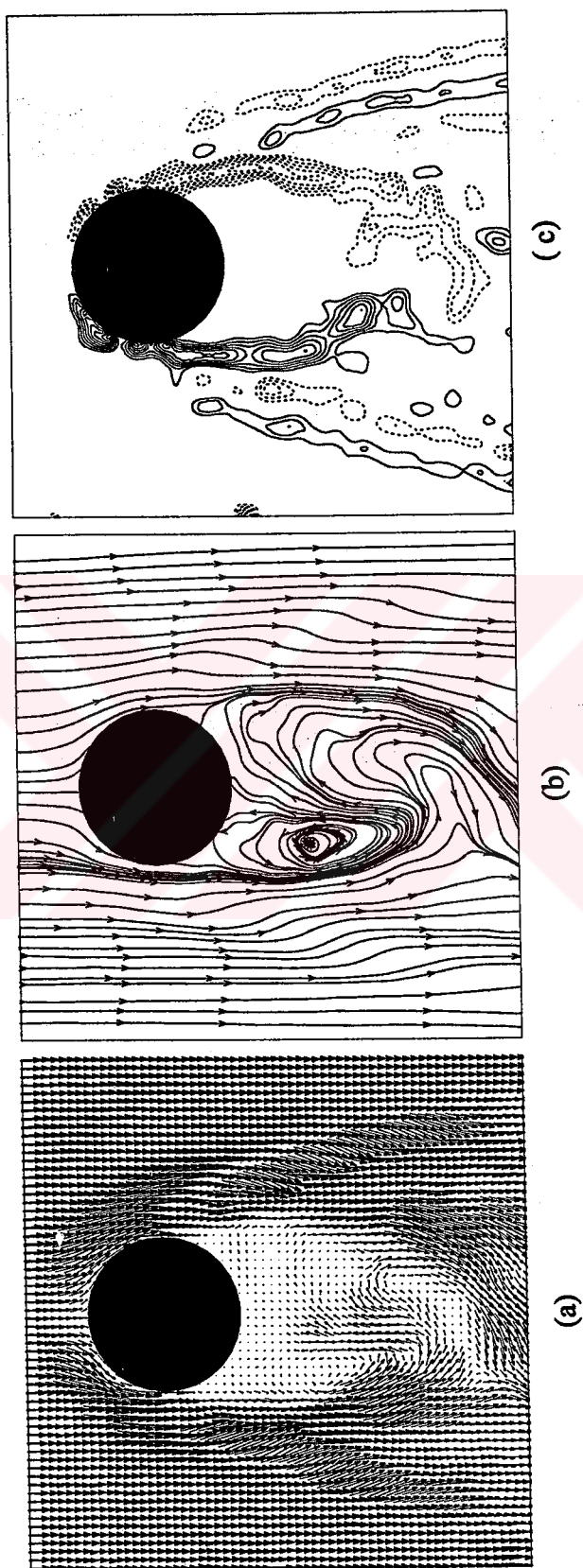


Figure 5.56. Experimental results obtained at the centerline of the flow passage for $F_s = 0.365$ at $Re = 4000$ at $N=8$, plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

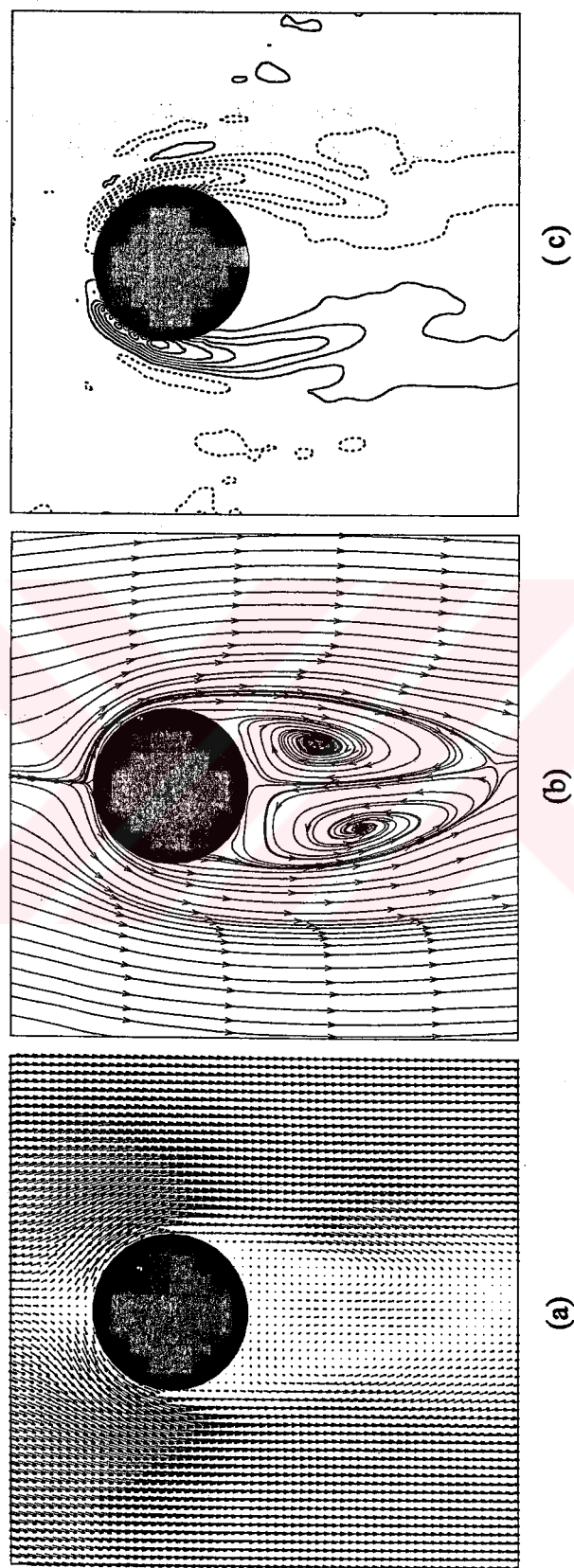


Figure 5.57. Experimental results obtained close to the fin surface for $Fr = 0.365$ at $Re = 600$, plan view
(a) Time averaged velocity vector field
(b) Time averaged streamline pattern
(c) Time averaged vorticity contours

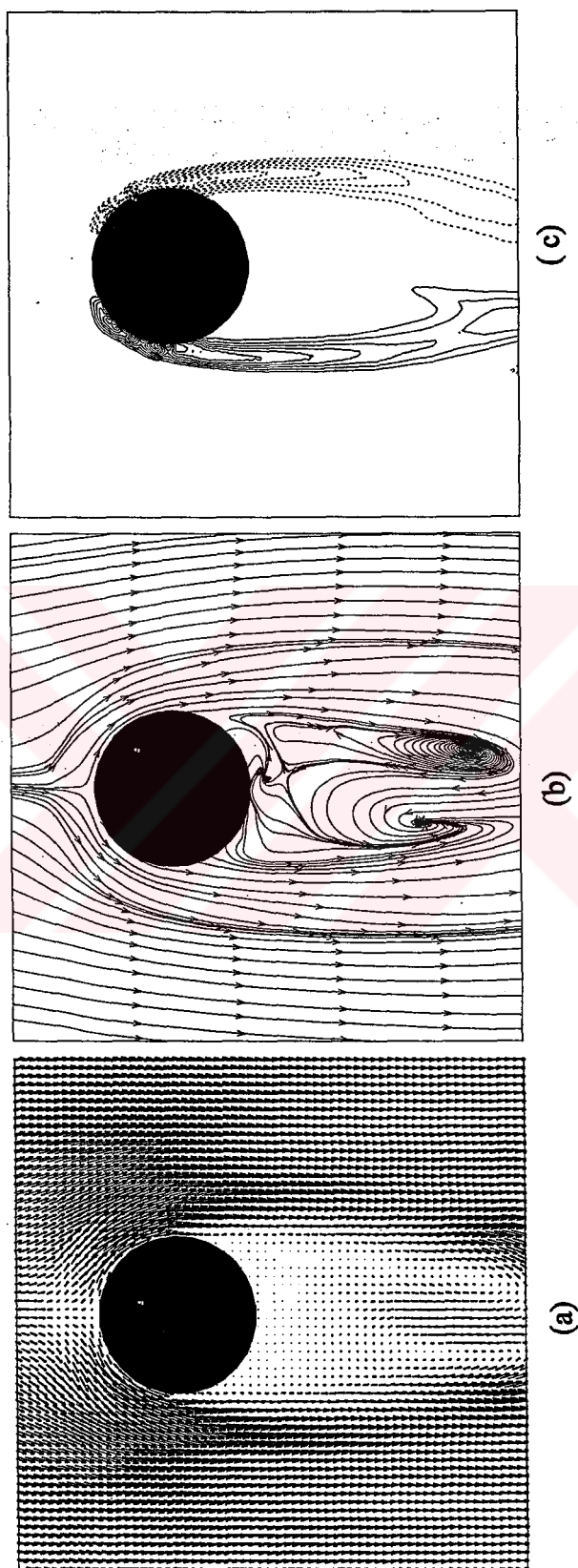


Figure 5.58. Experimental results obtained close to the fin surface for $Fr = 0.365$ at $Re = 1460$, plan view
(a) Time averaged velocity vector field
(b) Time averaged streamline pattern
(c) Time averaged vorticity contours

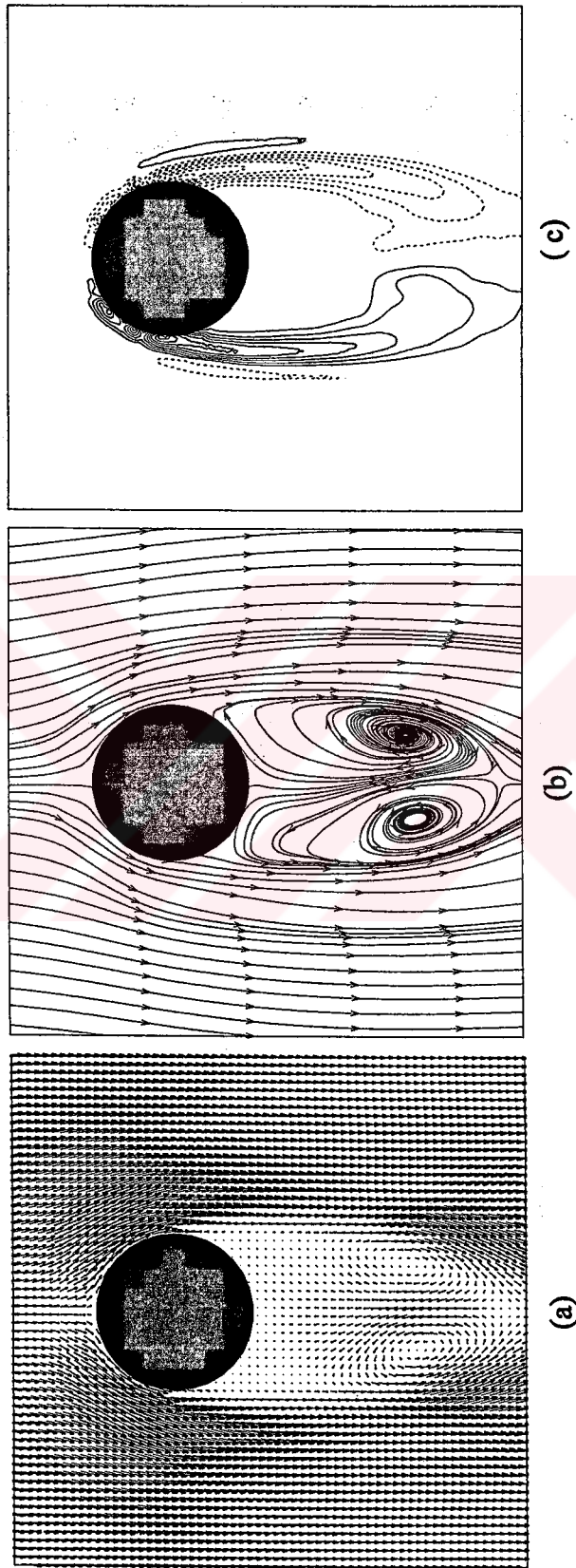


Figure 5.59. Experimental results obtained close to the fin surface for $Fr_s = 0.365$ at $Re = 4000$, plan view

(a) Time averaged velocity vector field
(b) Time averaged streamline pattern
(c) Time averaged vorticity contours

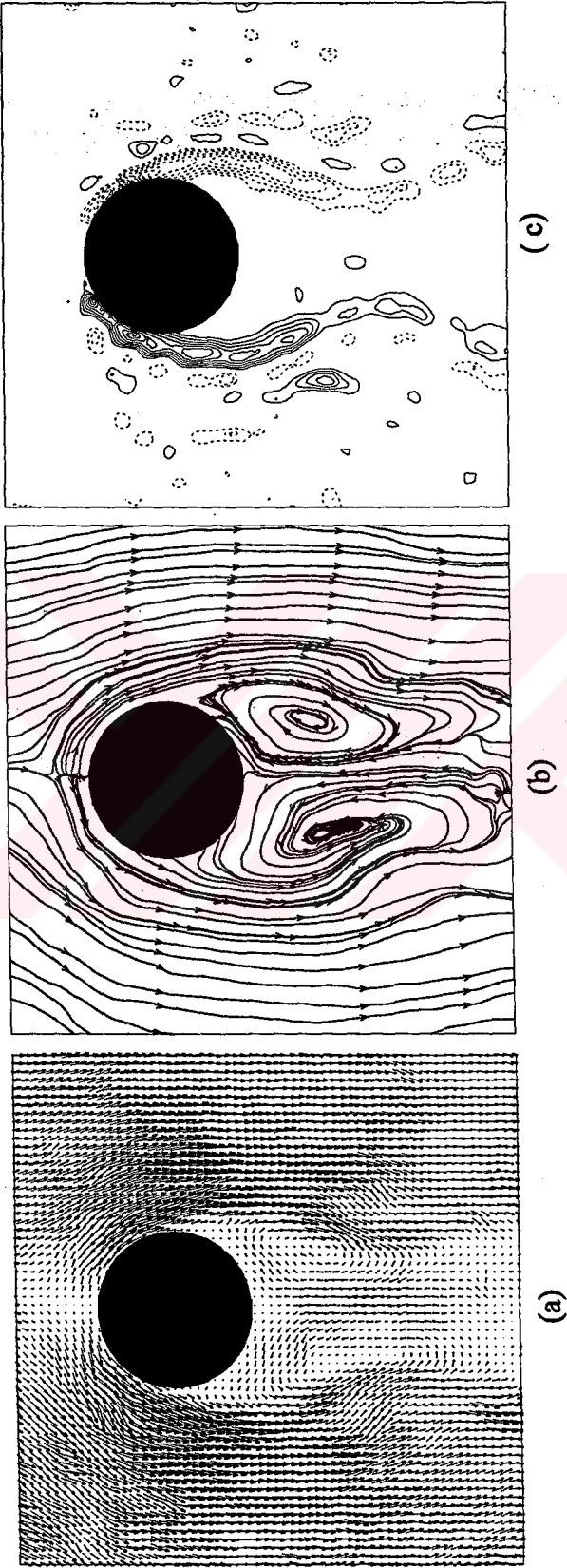


Figure 5.60. Experimental results obtained close to the fin surface for $Fr = 0.365$ at $Re = 600$ at $N = 6$, plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

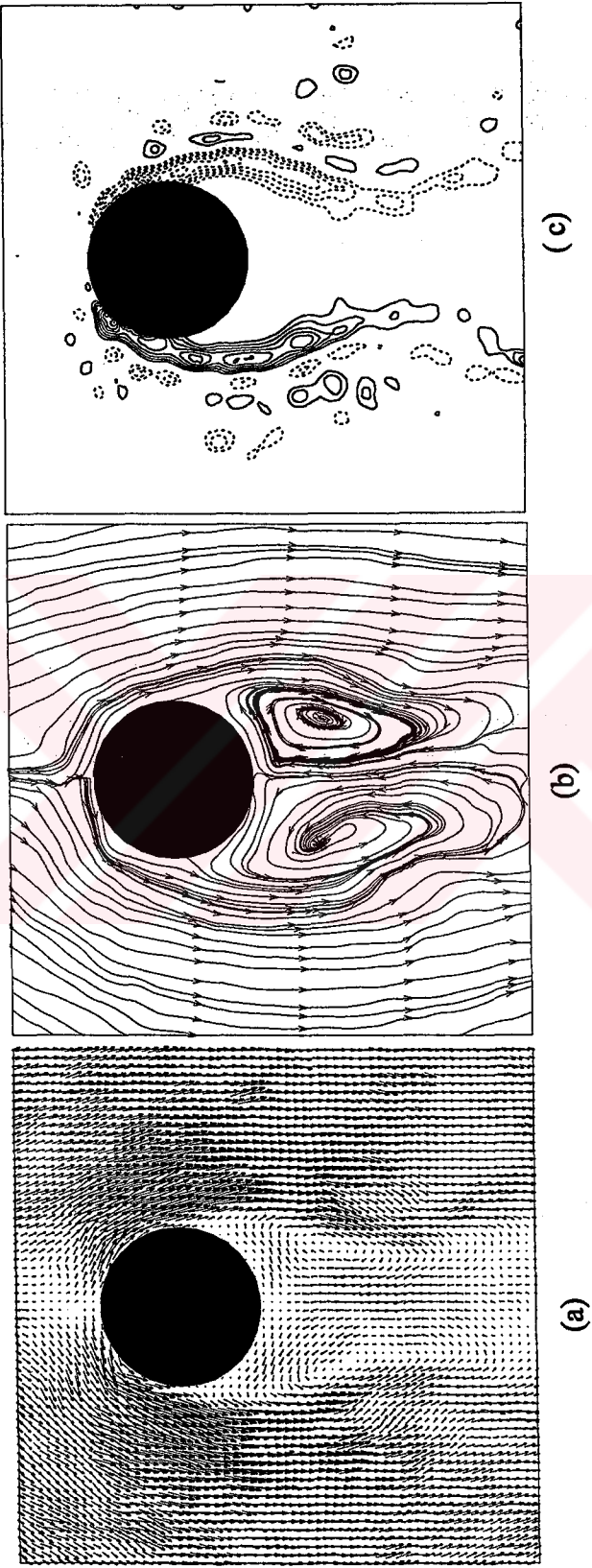


Figure 5.61. Experimental results obtained close to the fin surface for $Fr = 0.365$ at $Re = 600$ at $N = 8$, plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

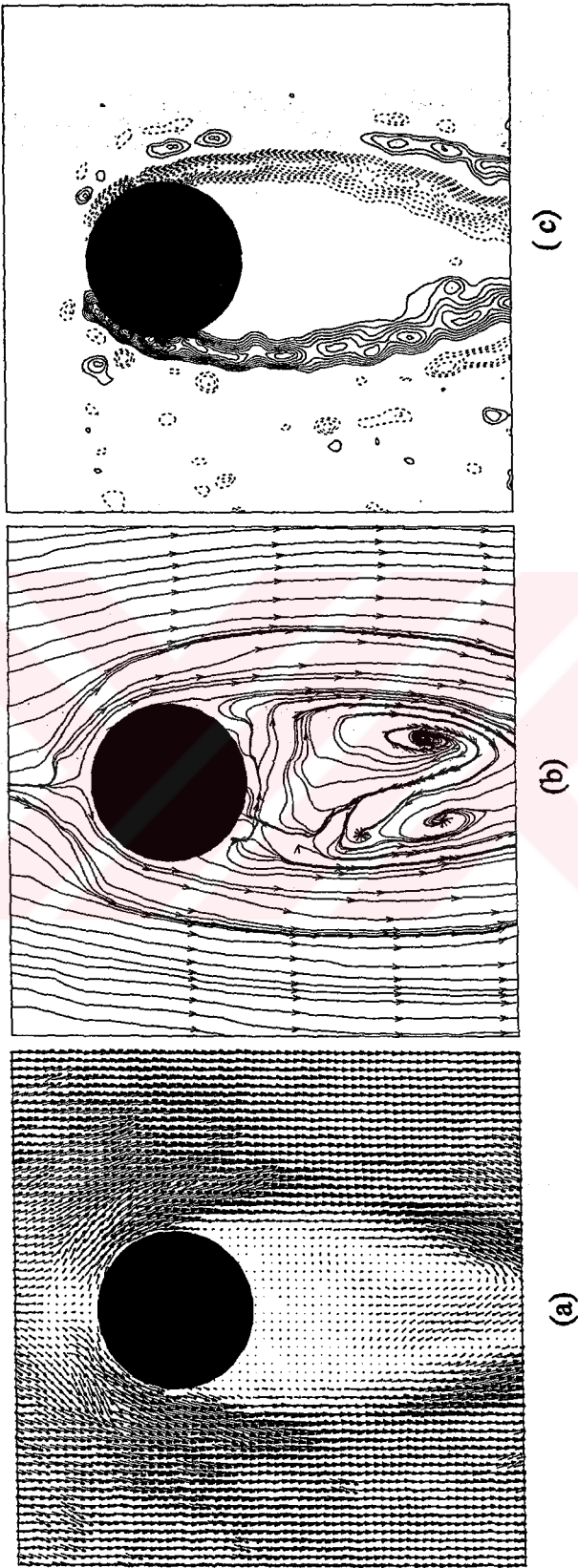


Figure 5.62. Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 1460$ at $N=6$, plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

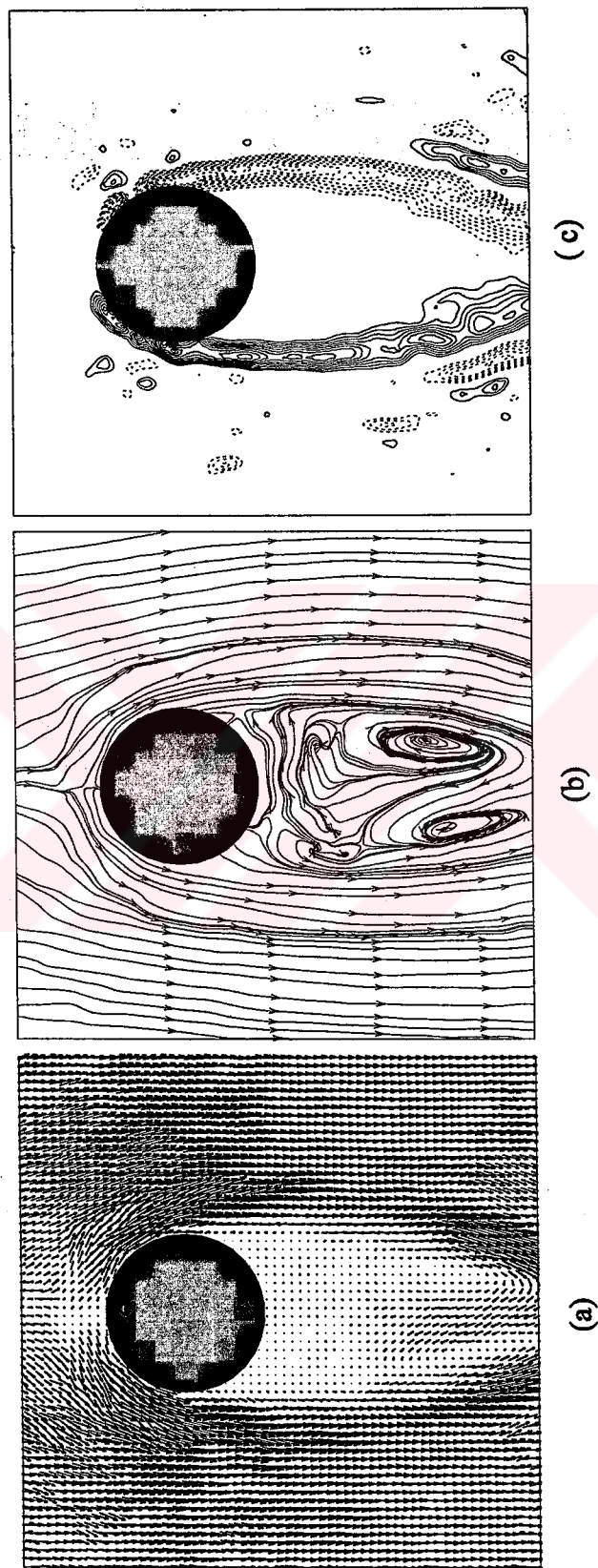


Figure 5.63. Experimental results obtained close to the fin surface for $F_s=0.365$ at $Re=1460$ at $N=8$, plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

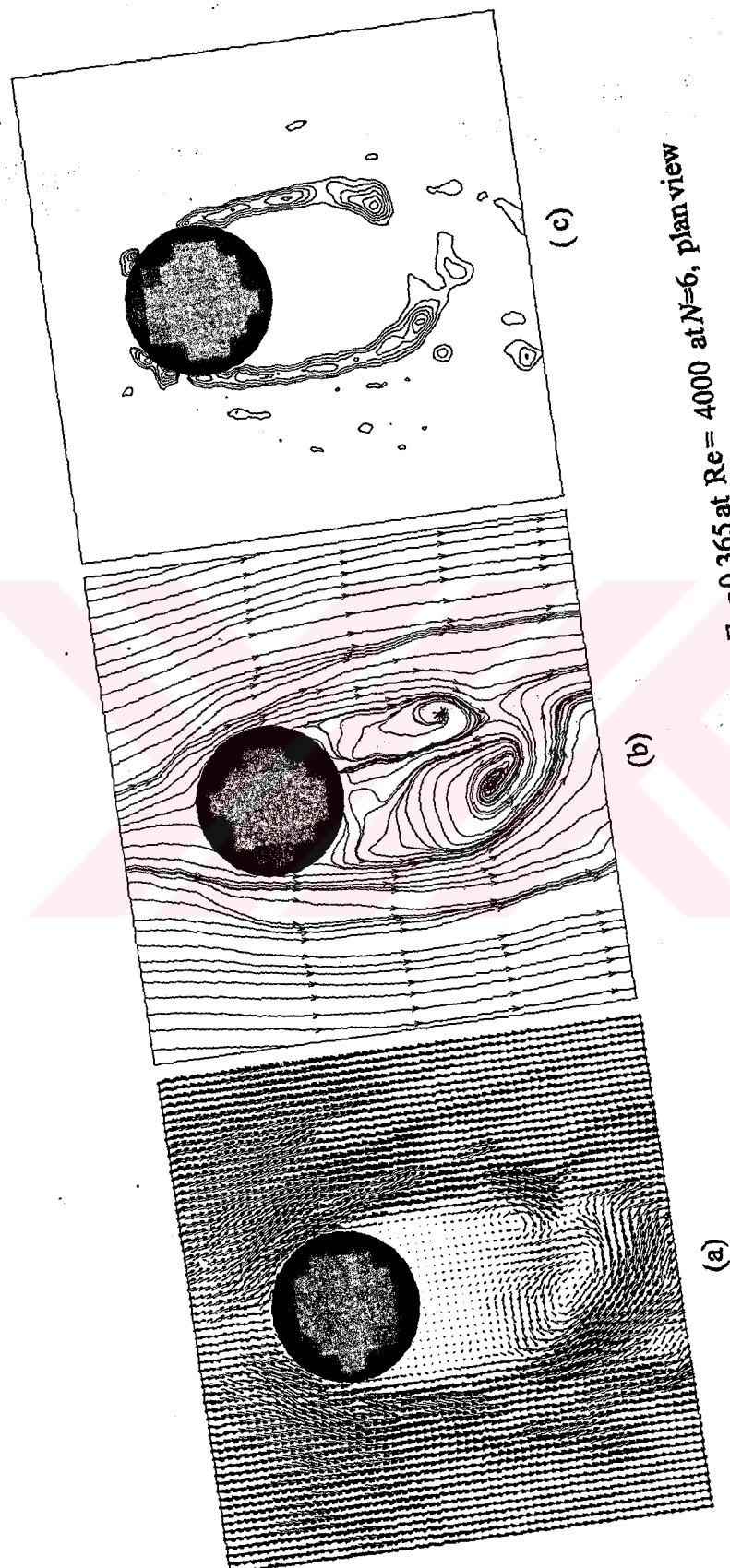


Figure 5.64. Experimental results obtained close to the fin surface for $F_s = 0.365$ at $Re = 4000$ at $N=6$, plan view
 (a) Instantaneous velocity vector field
 (b) Instantaneous streamline pattern
 (c) Instantaneous vorticity contours

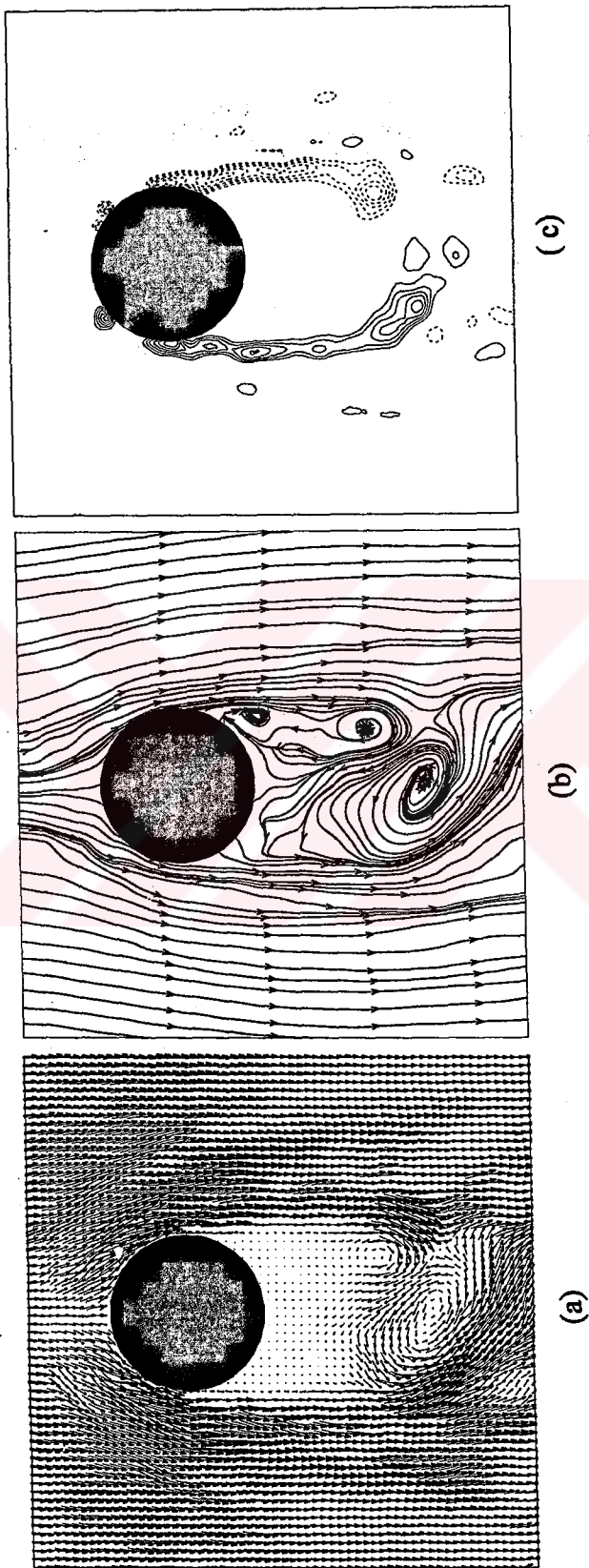


Figure 5.65. Experimental results obtained close to the fin surface for $Fr = 0.365$ at $Re = 4000$ at $N = 8$, plan view

(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

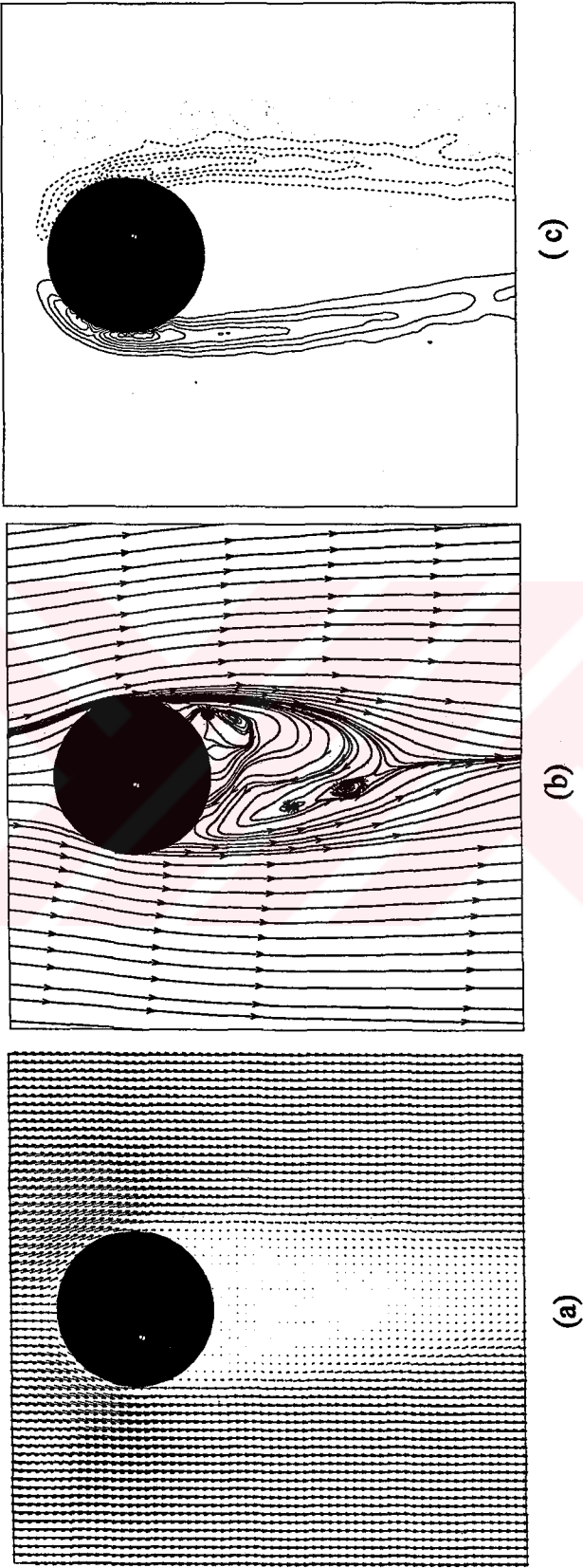


Figure 5.66. Experimental results obtained at the centerline of the flow passage for $Fr = 0.116$ at $Re = 1240$, plan view
(a) Time averaged velocity vector field
(b) Time averaged streamline pattern
(c) Time averaged vorticity contours

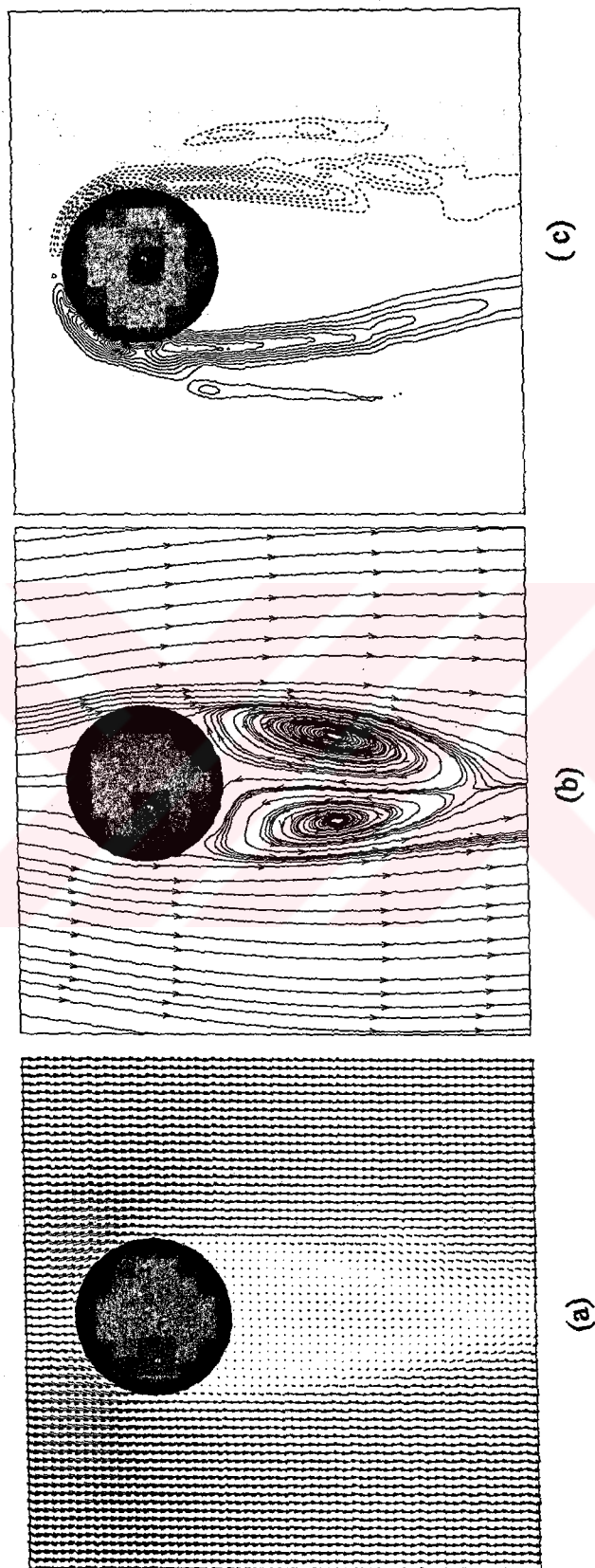


Figure 5.67. Experimental results obtained at the centerline of the flow passage for $F_s = 0.116$ at $Re = 2500$, plan view
 (a) Time-averaged velocity vector field
 (b) Time-averaged streamline pattern
 (c) Time-averaged vorticity contours

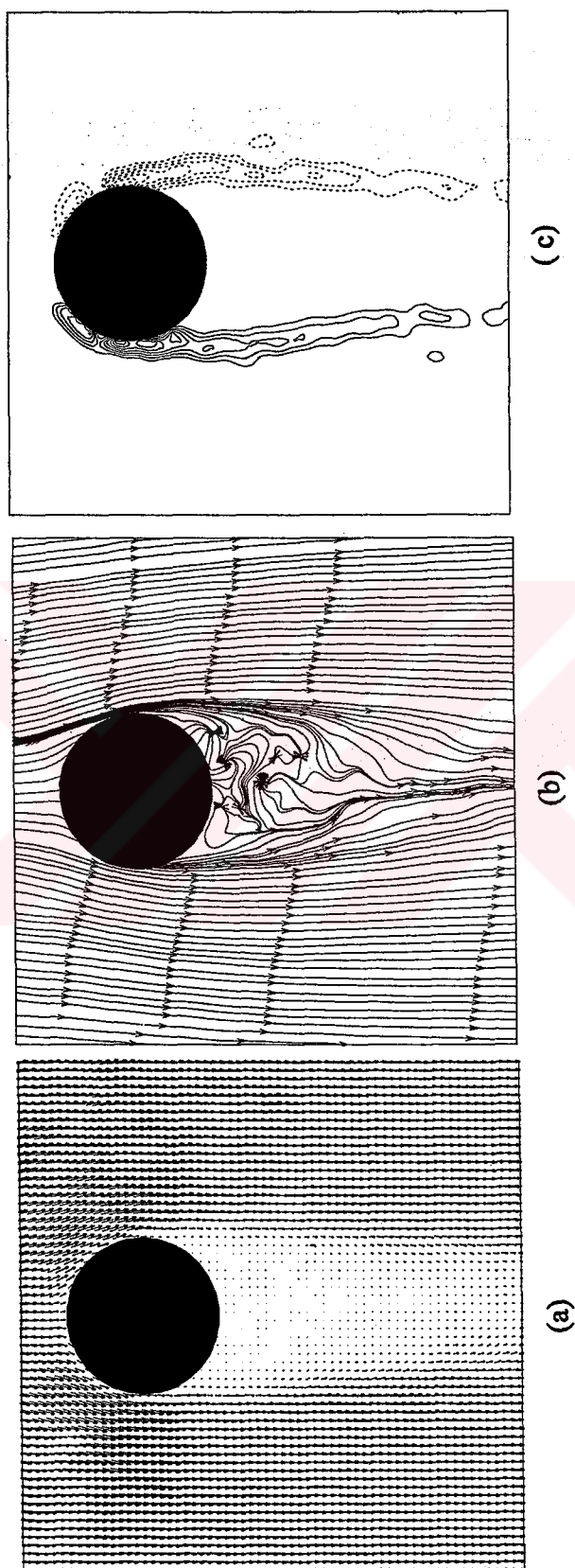


Figure 5.68. Experimental results obtained at the centerline of the flow passage for $F_s = 0.116$ at $Re = 1240$ at $N = 6$ plan view

(a) Instantaneous velocity vector field

(b) Instantaneous streamline pattern

(c) Instantaneous vorticity contours

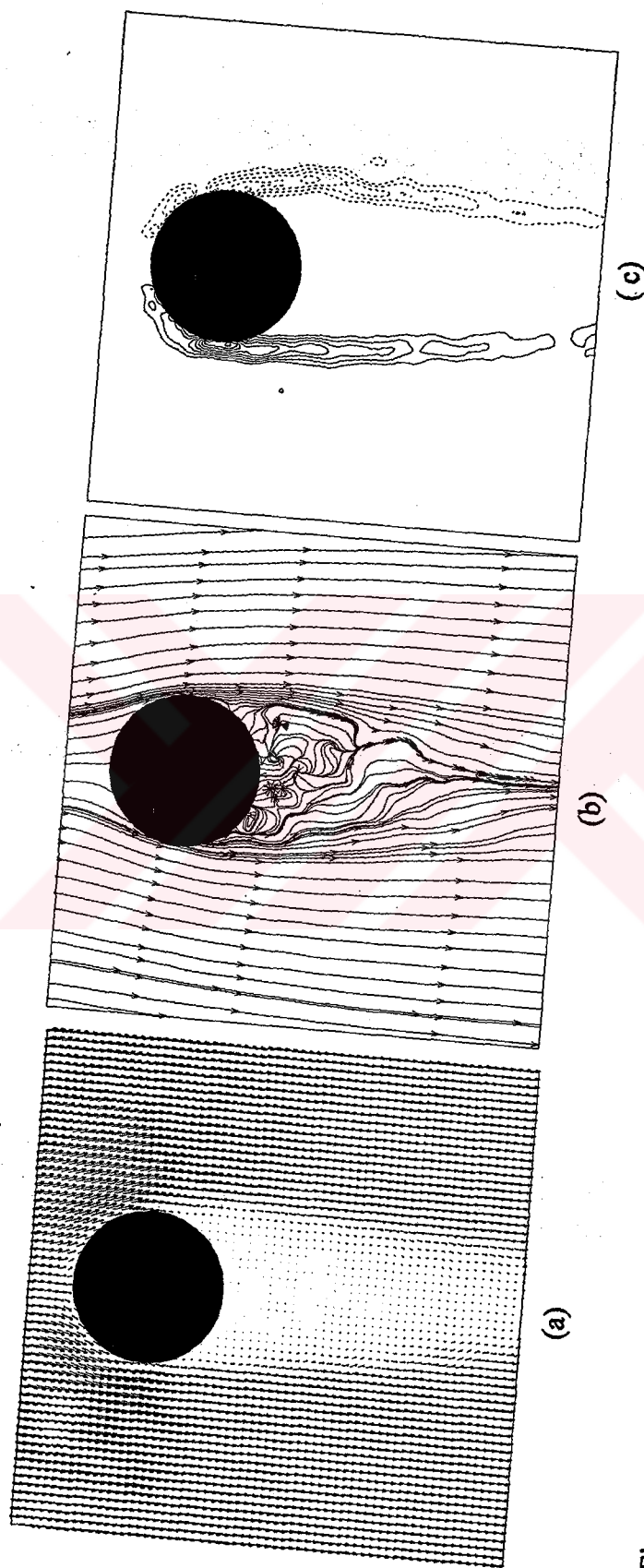


Figure 5.69. Experimental results obtained at the centerline of the flow passage for $F_s = 0.116$ at $Re = 1240$ at $N = 8$ plan view
 (a) Instantaneous velocity vector field
 (b) Instantaneous streamline pattern
 (c) Instantaneous vorticity contours

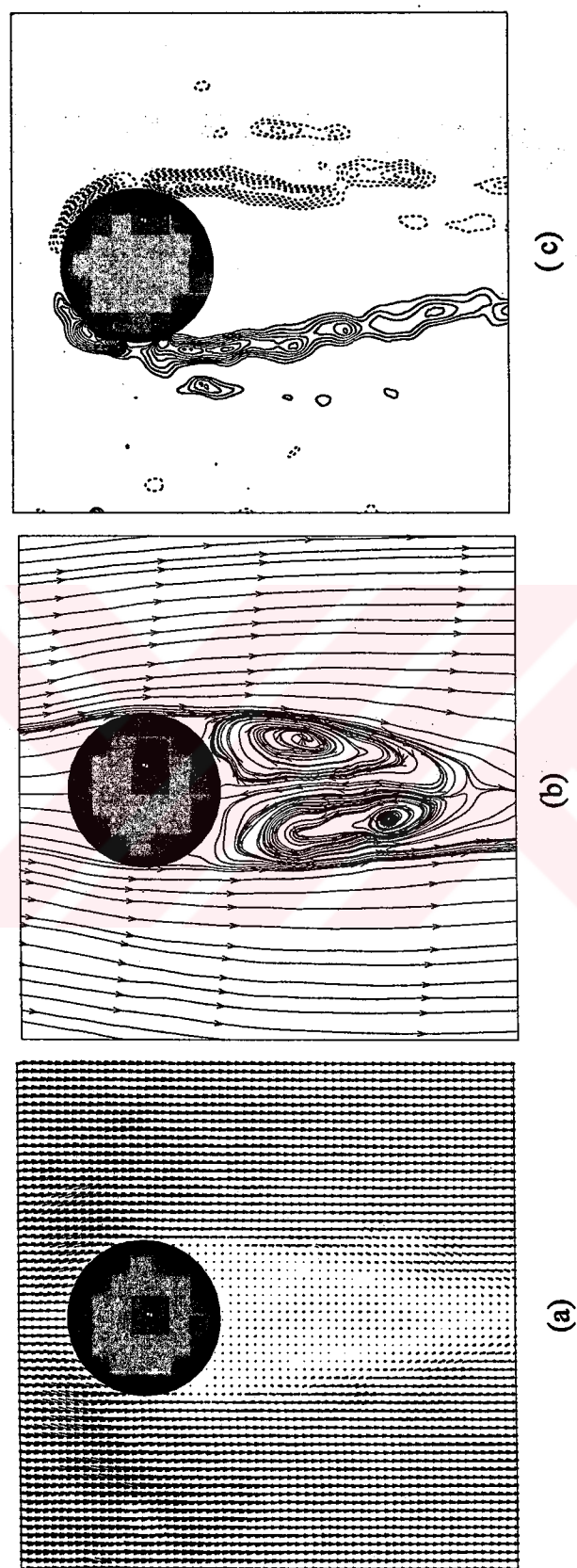


Figure 5.70. Experimental results obtained at the centerline of the flow passage for $Fs=0.116$ at $Re=2500$ at $N=6$ plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

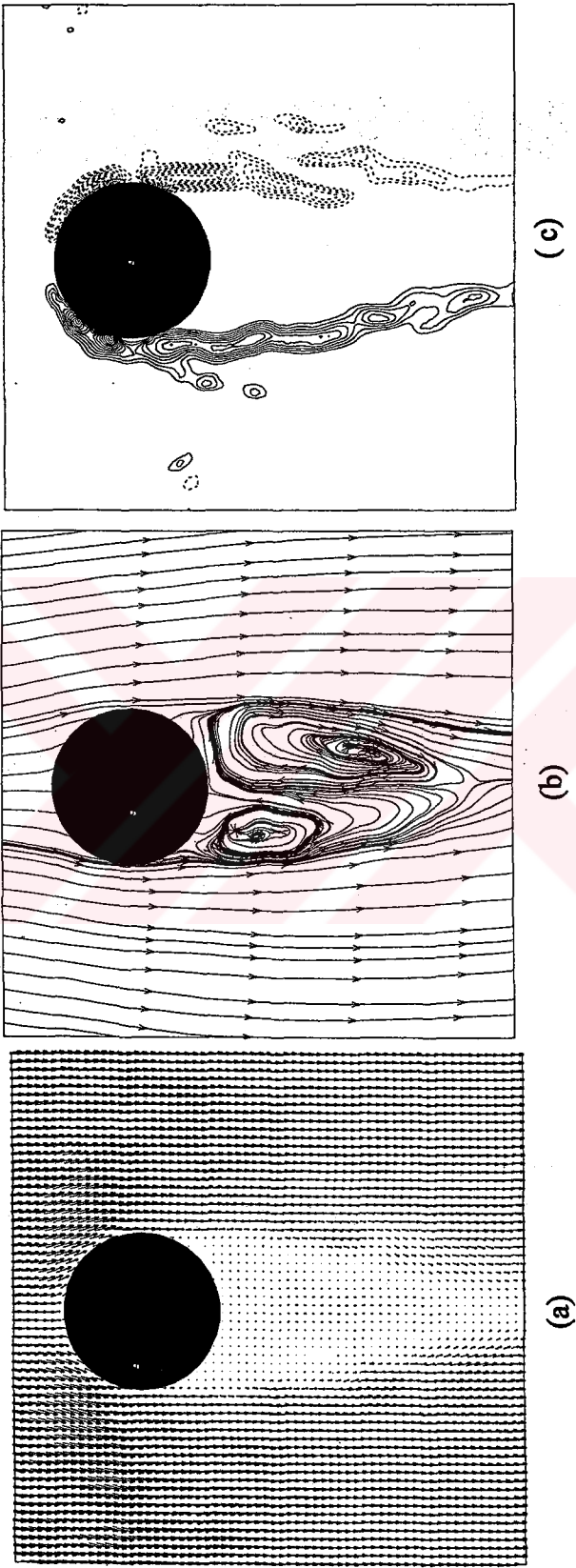


Figure 5.71. Experimental results obtained at the centerline of the flow passage for $F_s = 0.116$ at $Re = 2500$ at $N = 8$ plan view
(a) Instantaneous velocity vector field
(b) Instantaneous streamline pattern
(c) Instantaneous vorticity contours

6. CONCLUSIONS

Following conclusions can be derived from the present numerical study:

1. A test model geometry comprised of a single tube row surrounded by a fin plate configuration can be a basis for a multi row plate fin and tube heat exchanger in the context of optimization and design of plate fin and tube heat exchangers. Fluid flow and corresponding heat transfer characteristics of the single row heat exchanger geometry can be extended to investigate multi row plate fin and tube heat exchangers.
2. An unstructured mesh system containing non-uniformly distributed (228x46x20) grid points is fine enough to predict the flow features and corresponding heat transfer characteristics in the heat exchanger passage.
3. Fluid flow and corresponding heat transfer characteristics of plate fin and tube heat exchangers have unsteady state, time dependent and three dimensional nature. That's why it is necessary to use three dimensional and time dependent models for the numerical solution of the problem.
4. Fin spacing, tube row number, tube arrangement, fin configuration and Reynolds number are important governing design parameters for the plate fin and tube heat exchangers.
5. Larger recirculation lengths are obtained at higher Reynolds numbers behind the fourth row at the middle of the flow passage as well as at the plane very near to the fin surface. The smaller recirculation length is obtained from the RNG $k-\varepsilon$ model simulation compared to the laminar model simulation at $Re = 1500$.
6. Formation of horseshoe vortex system in the flow passage is dependent more on the fin spacing range rather than Reynolds number range considered in the present work.
7. The heat transfer coefficient increases with an increase of the fin spacing and the Reynolds number.

8. The magnitude of the pressure drop along the tube surface in the staggered arrangement is found to be higher than that in the inlined arrangement.
9. Similarly, for the staggered arrangement, higher heat transfer coefficients are obtained than those for the inlined arrangement.
10. Numerical solution of the plate fin and tube heat exchangers having plain and wavy fin configurations working under similar operating parameters and velocity boundary conditions of daily used heat exchanger machines demonstrated that the optimum heat exchanger model has a fin spacing in the $0.20 \leq F_s \leq 0.30$ range, wavy fin configuration, four tube row numbers with staggered layout.
11. Time evolution of the flow and heat transfer properties showed that increased local heat transfer can be attributable to the formation of horseshoe vortex in the forward part of the tube. On the other hand, decreased heat transfer coefficients obtained in the backward part of the tube can be attributed to the recirculation region and strong interaction of singular points.
12. As the Reynolds number is increased, turbulence effects on the fluid flow and heat transfer characteristics in the heat exchanger flow passage increase also. That's why it is necessary to employ turbulence modeling for the numerical solution of the flow and heat transfer problem for these type of heat exchanger passages.
13. Results obtained from Large Eddy Simulation (LES) method, which is based on filtered Navier Stokes Equations based turbulence models are in better agreement with the experimental data compared to those obtained from time averaged Navier Stokes Equations based turbulence models. Selection of the turbulence model directly influences the results of heat transfer and pressure drop.
14. The LES reproduces more compact and stronger horseshoe vortex mechanism compared to the RANS equations based turbulence models. The location of the horseshoe vortex for the LES is also different than the other turbulence models: It moves further downstream around the tube surface resulting in better heat transfer performance.

15. The dynamic LES method in which the Smagorinsky constant is varying with time and space proposed in the present investigation improves the numerical results when compared to the standard LES method.
16. Time evolution of the local flow structure gives a fundamental information on the heat transfer characteristics in the flow passage.
17. The size of the wake region differs depending on the Reynolds number and fin spacing. As the Reynolds number is increased, the size of the wake region becomes smaller.
18. From the instantaneous PIV data, strong singular point interactions are observed in the wake region. As the Reynolds number and fin spacing are increased, the formation and interaction of singular points decreases in the wake region. The formation and interaction of the singular points increase near the fin due to the presence of the fin surface.
19. The positive and the negative vorticity layers are approximately equal and symmetric with each other for larger fin spacings while the positive vorticity layers are longer and the negative vorticity layers are shorter for small fin spacings. For larger fin spacings, the vorticity contours become shorter and wider as the Reynolds number is increased.
20. The present numerical study is tested against several experimental studies available in the literature and captured convenient experimental results.
21. The results of the present numerical study are verified by the experimental study based on the PIV technique. It is shown that there is a strong agreement between the experimental and the numerical results.

7. FUTURE WORK

In the present study, laminar and turbulent heat transfer characteristics are investigated for the plate fin and tube heat exchangers. The flow behavior in the heat exchanger flow passage is really a complex, three-dimensional and unsteady with flow separation, reattachment, recirculation zones, and vortices in the wake of the tubes and the fins that affect both pressure drop and heat transfer characteristics. The numerical solution of this complex flow problem strongly depends on some factors such as boundary conditions, mesh sensitivity, material properties, convergence criterion, and most importantly, the computer hardware. Even the most powerful computers today, may not be sufficient to predict the flow structure for complex geometries. Therefore the numerical studies should be supported by experimental studies in which the velocity and temperature fields can be measured experimentally. In the present study, the experimental verification is made by using the PIV (Particle Image Velocimetry) technique where instantaneous velocity vector measurements are made for only one cylinder configuration. The present investigation can be extended to investigate the effect of inlined and staggered tube arrangements by using PIV technique. In addition to the measurement of velocity vector fields, the temperature fields may be also measured in order to obtain the instant temperature maps of the flow field. The instant temperature maps can be measured by an optical technique called Planar Laser-Induced Fluorescence (planar-LIF). By using this technique, the present study can be further extended including real temperature fields and corresponding heat transfer properties of plate fin and tube heat exchangers. In addition to the real temperature measurements, the optimization study considered in the present dissertation may be further extended to investigate effects of tube diameter, effects of tube shapes having oval or elliptical configurations, and effects of transverse and longitudinal distance between the tubes on the flow and heat transfer characteristics of plate fin and tube heat exchangers. Moreover, effects of vortex generators such as delta wings placed on the fins and between the tubes on the flow behavior can be also investigated in order to improve the heat transfer characteristics of plate fin and tube heat exchangers.

REFERENCES

- ABBOT, M.B., BASCO, B.R., 1989. Computational Fluid Dynamics- An Introduction for Engineers. Longman Scientific & Technical Harlow, UK.
- ANDERSON, J.D., 1995. Computational Fluid Dynamics. The Basics With Applications. ISBN 0-07-001685-2, McGraww hill Int. Eds., Singapore, 547 p.
- AKAR, M.A., 2003. Turbulent Wakes Generated by side-by-Side Cylinders in Shallow Water. MSc Thesis, Cukurova University, Institute Of Natural and Applied Sciences, Department of Mechanical Engineering, Adana, 83 p.
- AY, H., JANG, J.Y., YEH, J.N., 2002. Local Heat Transfer Measurements of Plate Finned- Tube Heat Exchangers by Infrared Thermography. Int. J. Heat Mass Transfer, 45: 4069-4078.
- BALLIO, F., GUADAGNINI, A., MALAVASI, S., 1998. Stresses Due to a Horseshoe Vortex at a Surface- Cylinder Intersection. International Communications in Heat Exchanger Symposium (Poster Session, 31.08.1998, 12:00-12:15)
- BARSAMIAN, H.R., HASSAN, Y.A., 1997. Large Eddy Simulation of Turbulent Crossflow in tube Bundles. Nuclear Engineering and Design, 172:103-122.
- BEALE, S.B., SPALDING, D.B., 1999. . A Numerical Study of Unsteady Fluid flow in In- Line and Staggered Tube Banks. Journal of Fluids and Structures, 13:723-754.
- BRANDT, T., 2004. Turbulence Simulation (DNS and LES). (from internet: http://www.aero.hut.fi/Englanniski/CFD_text/turbulence_simulation.html)
- CHEN, C.C., WUNG, T.S., 1989. Finite Analytic Solution of Convective Heat Transfer for Tube Arrays in Crossflow: Part II- Heat Transfer Analysis. Journal of Heat Transfer, August, 111: 641-648.
- CIOFALO, M., STASIEK, J., COLLINS, M.W., 1996. Investigation of Flow and Heat Transfer in Corrugated Passages –II. Numerical Results. Int. J. Heat and Mass Transfer, 39: 165-192.
- CRITOPH, R.E., HOLLAND, M.K., FISHER, M., 1999. Comparison of Steady State and Trasnsient Methods for Measurement of Local Heat Transfer in Plate

- fin- Tube Heat Exchangers Using Liquid Cristal Thermography With Radiant Heating. *Int. J. Heat and Mass Transfer*, 42: 1-12.
- DEARDORFF, J.W., 1970. A Numerical Study of Three Dimensional Turbulent Channel Flow at Large Reynolds Numbers. *Journal of Fluid Mechanics*, 41: 453-480.
- ECHIGO, R., YOSHIDA, H., HANAMURA, K., MORI, H., 1992. Fine- Tube Heat Exchanger Woven With Threads. *Int. J. Heat Mass Transfer*, Pergamon Press Plc, 35:(3), 711-717.
- FLUENT DYNAMICS SOFTWARE, FLUENT, 1998.
- FOTHERBY, L.M., RUFF, J.F., 1995. Bridge Scour Protection Using Toskones, Phase 1. Pennsylvania Dept. Of Transportation, Report No: 91-02, June.
- GOLDSTEIN, L.G., SPARROW, E.M., 1976. Experiments on the Transfer Characteristics of a Corrugated Fin and Tube Heat Exchanger Configuration. *J. Heat Transfer*, Trans ASME, February: 26-32.
- HALICI, F., TAYMAZ, I., GUNDUZ, M., 2001. The Effect of the Number of Tube Rows on Heat, Mass, and Momentum Transfer in Flat- Plate Finned Tube Heat Exchangers. *Energy*, 26: 963-972.
- HOUGHTON, J.M., INGHAM, D.B., HEGGS, P.J., 1992. The One- Dimensional Analysis of Oscillatory Heat Transfer in a Fin Assembly. *Journal of Heat Transfer*, Transactions of the ASME, AUGUST, 114: 548-552.
- LÜBCKE, H., SCHMIDT, St., RUNG, F., THIELE, F., 2001. Comparison of LES and RANS in Bluff- Body Flows. *Journal of Wind Engineering*, 89: 1471-1485.
- JALURIA, Y., TORRANCE, K.E., 1986. *Computational Heat Transfer*. Hemisphere Publishing Corporation, ISBN 0-89116-286-0, ISSN 0272-4804, 366 p.
- JANG, J.Y., CHEN, L.K., 1997. Numerical Analysis of Heat Transfer and Fluid Flow in a Three- Dimensional Wavy Fin and Tube Heat Exchanger. *International Journal of Heat and Mass Transfer*, 40(16): 3981-3990.
- JANG, J.Y., WU, M.C., CHANG, W.J., 1996. Numerical and Experimental Studies of Three- Dimensional Plate Fin and Tube Heat Exchangers. *International Journal of Heat and Mass Transfer*, 39(14): 3057-3066.

- KAKAC, S., LIU, H., 1997. Heat Exchangers, Selection, Rating, and Thermal Design, CRC Press, 432 p.
- KAKAC, S., LIU, H., SABOYA, F.E.M., SPARROW, E.M., 1974. Local and Average Transfer Coefficients for One Row Plate Fin and Tube Heat Exchanger Configurations. J. Heat Transfer, Trans ASME, August: 265-272.
- KAVIANY, M., RECKKER, M., 1990. Performance of a Heat Exchanger Based on Enhanced Heat diffusion in Fluids by Oscillation: Experiment. Journal of Heat Transfer, Transactions of the ASME, FEBRUARY 1990, 112: 56- 63.
- KIM, H.H., YOUN, B., WEBB, R.L., 1999. Air- side Heat transfer and Friction Correlations for Plain fin- and- tube Heat Exchangers With Staggered Tube Arrangements. Journal of Heat Transfer, Transactions of the ASME, 121: 662-667.
- KIM, J.Y., SONG, T.H., 2002. Microscopic Phenomena and Macroscopic Evaluation of Heat Transfer From Plate Fins/Circular tube Assembly Using Naphthalene Sublimation Technique. International Journal of Heat and Mass Transfer, 45: 3397-3404.
- KIM, J.Y., SONG, T.H., 2003. Effect of Tube Allignmnet an the Heat/Mass Transfer From a Plate Fin and Two- Tube Assembly: Naphthalene Sublimation Results. International Journal of Heat and Mass Transfer, 46: 3051-3059.
- KIM, N.H., YUN, J.H., WEBB, R.L., 1997. Heat Transfer and Friction Correlations for Wavy Plate Fin- and- Tube Heat Exchangers. Journal of Heat Transfer, Transactions of the ASME, 119: 560-567.
- KUNDU, D., SKEIKH, A.H., LOU, D.Y.S., 1991a. Heat Transfer Predictions in Cross Flow Over Cylinders Between Two Parallel Plates. Numerical Heat Transfer, Part A, 19: 361-377.
- KUNDU, D., SKEIKH, A.H., LOU, D.Y.S., 1991b. Pressure and Heat Transfer in Crossflow Over Cylinders Between Two Parallel Plates. Numerical Heat Transfer, Part A, 19: 345-360.
- KUNDU, D., SKEIKH, A.H., LOU, D.Y.S., 1992. Heat Transfer in Crossflow Over Cylinders Between Two Parallel Plates. Numerical Heat Transfer, Part A, 114: 558-564.

- KUPPAN, T., 2000. Heat Exchanger Design Handbook, Marcell Dekker, Inc., New York, Basel, ISBN: 0-8247-9787-6, 1119 p.
- LAUNDER, B.E., SPALDING, D.B., 1972. Mathematical Models of Turbulence. Academic Press, London.
- LILLY, D.K., 1966. On the Application of Eddy Viscosity Concept in the Inertial Sub-range of Turbulence., NCAR Manuscript 123.
- LESIEUR, M., 1997. Turbulence in Fluids., Kluwer Academic Publishers., 3rd edition, Netherlands, ISBN: 0-7923-4415-4, 515.
- LOZZA, G., MERLO, U., 2001. An Experimental Investigation of Heat Transfer and Friction Losses of Interrupted and Wavy Fins for Fin-and-Tube Heat Exchangers. Int. J. Refrigeration, 24: 409-416.
- MADI, M.A., JOHNS, R.A., HEIKAL, M.R., 1998. Performance Characteristics Correlation for Round Tube and Plate Finned Heat Exchangers. Int. J. Refrigeration, 21(7):507-517.
- MAJUMDAR S., RODI, W., 1985. Numerical Calculation of Turbulent Flow Past Circular Cylinder. Proceedings of the 3rd Symposium on Physical Aspects of Aerodynamic Flows, Long Beach, CA.
- MARTIN, J.J., 2004. Flow Visualization Using Particle Image Velocimetry. (<http://www.me.uvic.ca>), 3p.
- MENDEZ, R.R., SEN, M., YANG, K.T., MCCLAIN, R., 2000. Effect of Fin Spacing on Convection in a Plate Fin and Tube Heat Exchanger. Int. J. Heat Mass Transfer, 43: 39-51.
- MILLER, W., Jr., 2003. Model for the Time Rate of Local Sediment Scour at a Cylindrical Structure. PhD Thesis, Graduate School of University of Florida.
- MISAG-147, 2003. Jeotermal Enerji ile Isıtılan Seralardaki Isı Değiştiricilerinin Tasarımında Türbülanslı Isı Taşınımının Bilgisayarlı Modellenmesi, TUBITAK, Project No: MISAG-147, Final Report, 70p.
- MUZZAMMIL, M., GANGADHARIAH, T., 2003. The Mean Characteristics of Horseshoe Vortex at a Cylindrical Pier. Journal of Hydraulic Research, 41(3): 285-297.

- PEYRET, R., 1996. Handbook of Computational Fluid Mechanics. Academic Press Limited, France, ISBN: 0-12-553010-2, 467 p.
- RABBIT, M.J., 1997. Some Validation of Standard, Modified and Non- Linear $k - \epsilon$ Turbulence Models. Int. J. For Numerical Methods in Fluids. 24:965-986.
- REYNOLDS, W.C., 1987. Fundamentals of Turbulence for Turbulence Modelling and Simulation. Lecture Notes for Von Karman Institute Agard Report No: 755.
- RICHARDSON, E.V., DAVIS, S.R., 1995. Evaluating Scour at Bridges. U.S. Federal Highway Administration Hydraulic engineering Circular No. 18, Publication No. FHWA-IP-90-017, Third Edition, November.
- ROCHA, L.A.O., SABOYA, F.E.M., VARGAS, J.V.C., 1997. A Comparative Study of Elliptical and Circular Sections in One- and Two- Row Tubes and Plate Fin Heat Exchangers. International Journal of Heat and Fluid Flow, 18: 247-252.
- ROHSENOW, W.M., HARTNETT, J.P., CHO, Y.I., 1998. Handbook of Heat Transfer. Third Edition. McGraw Hill, ISBN 0-07-053555-8, New York, 1330p.
- ROSMAN, E.C., CARAJILESCOV, P., SABOYA, F.E.M., 1984. Performance of One- and Two- Row Tube and Plate Fin Heat Exchanger. J. Heat Transfer, Trans ASME, 106: 627-632.
- SABOYA, F.E.M., SPARROW, E.M., 1974. Local and Average Transfer Coefficients for One Row Plate Fin and Tube Heat Exchanger Configurations. J. Heat Transfer, Trans. ASME., August: 265-272.
- SCHUMMAN, U., 1973. Ein Untersuchung Uber der Berechnung der Turbulent Stromungen im Platten und Ringspalt-Kanelen. Dissertation, Karlsruhe.
- SESHIMO, Y., FUJII, M., 1991. . An Experimental Study on the Performance of Plate fin and Tube Heat Exchangers at Low Reynolds Numbers. ASME/JSME Thermal Engineering Proceedings, (4): 449-454.
- SHAH, R.K., HEIKAL, M.R., THONON, B., 1997. Advances in Numerical Analysis of Fluid Flow, Heat Transfer, and Flow Friction Characteristics of Compact Heat Exchanger Surfaces. Proceedings. Int. Symposium on Fluid Flow and Heat Transfer, Ankara, Turkey, 68-87.

- SHEPPARD, D., Prof. Dr., 2004. Course Notes of the Lecture "EOC 6430 Coastal Structures", University of Florida, Civil and Coastal Engineering Department, (from internet).
- SHERWIN, K., HORSLEY, M., 1996. Thermofluids. Chapman and Hall, ISBN 0412598000.
- SHIH, T.H., LIOU, W.W., SHABBAR, A., ZHU, J., 1995. A New $k-\varepsilon$ Eddy Viscosity Model for High Reynolds Number Turbulent Flows- Model Development and Validation. Computers and Fluids, 24(3):227-238.
- SHIH, C., 1999. Quantitative Flow Visualization. Dept. Of Mechanical Eng., Florida State University, Course Notes, (from internet).
- SMAGORINSKY, J.S., 1963. General Circulation Experiments With the Primitive Equations: Part 1, Basic Experiments. Monthly Weather Review, 91: 99-164.
- SPEZIALE, C.G., 1987. On Nonlinear $k-l$ and $k-\varepsilon$ Models of Turbulence. J. Fluid Mechanics, 178: 459-475.
- TSAI, S.F., SHEU, T.W.H., 1998. Some Physical Insights into a Two- Row Finned Tube Heat Transfer. Computers and Fluids, 27(1): 29-46.
- TSAI, S.F., SHEU, T.W.H., 1999. Heat Transfer in a Conjugate Heat Exchanger With a Wavy Fin Surface. Int. J. Heat Mass Transfer, 42: 1735-1745.
- TUTAR, M., AKKOCA, A., 2004. Numerical Analysis of Fluid Flow and Heat Transfer Characteristics in Three Dimensional Plate Fin and Tube Heat Exchangers. Numerical Heat Transfer, Part A, 46: 1-21.
- TZANETAKIS, N., SCOTT, K., TAAMA, W.M., JACHUCK, R.J.J. 2004. Mass Transfer Characteristics of Corrugated Surfaces. Applied Thermal Engineering, 24: In Press.
- VALENCIA, A., FIEBIG, M., MITRA, N.K., 1996. Heat Transfer Enhancement by Longitudinal Vortices in a Fin- Tube Heat Exchanger Element With Flat Tubes. Journal of Heat Transfer, 118: 209-211.
- VAN DRIEST, E.R., 1956. On Turbulent Flow Near a Wall. Journal of Aerospace Science, 23(11): 1007-1011.

- VERSTEEG, H.K., MALALASEKERA, W., 1995. An Introduction to Computational Fluid Dynamics. ISBN-0-582-21884-5, Longman Group Ltd., Malaysia, 257 p.
- WACLAWICZ, K., 2001. The Investigation of Crossflow Velocity and Off-the-Surface Streamtrace Topology for a Moderately Swept Wing at Transonic Mach Numbers. MSc Thesis, Faculty of Virginia Polytechnic Institute and state University, Aerospace Engineering, Blacksburg, Virginia.
- WANG, C.C., 1999c. On the Air Side Performance of Fin and tube Heat Exchangers. Heat Transfer Enhancement of Heat Exchangers, Kakac et al. (eds), 141-162.
- WANG, C.C., CHANG, J.Y., CHIOU, N.F., 1999b. Technical Note: A Heat Transfer and Friction Correlation for Wavy Fin- and- tube Heat Exchangers. Int. J. Heat and Mass Transfer, 42:1919-1924.
- WANG, C.C., CHANG, J.Y., CHIOU, N.F., 1999d. Effects of Waffle Height on the Air Side Performance of Wavy fin- and- Tube Heat Exchangers, Heat transfer Engineering, 20(3): 45-56.
- WANG, C.C., CHI, K.Y., CHANG, C.J., 2000a. Heat Transfer and Friction Characteristics of Plain Fin-and-tube Heat Exchangers, Part II: Correlation. Int. J. Heat and Mass Transfer, 43: 2681-2691.
- WANG, C.C., CHI, K.Y., CHANG, C.J., 2000b. Heat Transfer and Friction Characteristics of Plain Fin-and-tube Heat Exchangers, Part I: New Experimental Data. Int. J. Heat and Mass Transfer, 43: 2693-2700.
- WANG, C.C., FU, W.L., CHANG, C.T., 1997. Heat Transfer and Friction Characteristics of Typical Wavy Fin- and- tube Heat Exchangers. Experimental Thermal and Fluid Science, 14: 174-186.
- WANG, C.C., HWANG, Y.M., LIN, Y.T., 2002. Empirical Correlations for Heat Transfer and Flow Friction Characteristics of Herringbone Wavy Fin-and-Tube Heat Exchangers. Int. J. Refrigeration, 25: 673-680.
- WANG, C.C., LIN, Y.T., LEE, C.J., CHANG, J.Y., 1999a. Investigation of Wavy Fin- and- tube Heat Exchangers: A contribution to Databank. Experimental Heat Transfer, 12:73-89.

- WANG, C.C., TSAI, Y.M., LU, D.C., 1998. Comprehensive Study of Convex-Louver and Wavy Fin- and Tube Heat Exchangers. *Journal of Thermophysics and Heat transfer*, 12(3): 423-430.
- WANG, C.C., WEBB, R.L., CHI, K.Y., 2000c. Data Reduction for Air- Side Performance of fin and tube Heat Exchangers. *Experimental Thermal and Fluid Science*, 21: 218-226.
- WIERBOWSKI, M., STASIEK, J., 2002. Liquid Crystal Technique for Heat transfer Investigation in a Fin- Tube Heat Exchanger Element. *Experimental Thermal and Fluid Science*, 26(2-4): 319-323.
- WILCOX, D.C., RUBESIN, M.W., 1980. Progress in the turbulence for complex flow fields including the effect of compressibility. NASA TP 1517.
- WUNG, T.S., CHEN, C.J., 1989. Finite Analytic Solution of Convective Heat Transfer for Tube Arrays in Crossflow: Part I- Flow Field Analysis, *Journal of Heat Transfer*, August, 111: 633-640.
- XING, Z., BAILI, Z., 2004. Study of Flow Around a Surface Mounted Cubical Obstacle Placed in a Channel. Fluid Dynamics Programme, Institution of High Performance Computing (From Internet).
- YAKHOT, A., ORSZAG, S.A., YAKHOT, V., ISRAELI, M., 1989. Renormalization Group Formulation of Large-Eddy Simulation. *Journal of Scientific Computing*, 4:139-158.
- YAKHOT, V., ORSZAG, S.A., THANGHAM, S., GATSKI, T.B., SPEZIALE, C.G., 1992. Development of Turbulence Models for Shear Flows by a Double Expansion Technique. *Physics of Fluids*, 4(7):1540-1520.
- YAN, W.M., SHEEN, P.J., 2000. Heat Transfer and Friction Characteristics of Fin and Tube Heat Exchangers. *Int. J. Heat Mass Transfer*, 43: 1651-1659.
- YOSHIZAWA, A., 1982. A Statistically- Derived Subgrid Scale Model for the Large- Eddy Simulation of Turbulence. *Phys. Fluids*, 25: 1532-1537
- ZDRAVITSCH, F., FLETCHER, C.A.J., BEHNIA, M., 1994. Laminar and Turbulent Heat Transfer Predictions in Tube Banks in Cross Flow. *Proceedings, International Conference on Fluid and Thermal Energy Conversion, Indonesia*, 12-16.

ZHANG, J., KUNDU, J., MANGLIK, R.M., 2004. Effect of Fin Waviness and Spacing on the Lateral Vortex Structure and Laminar Heat Transfer in Wavy-Plate-Fin Cores. *Int. J. Heat Mass Transfer*, 47:1719-1730.

ZUKAUSKAS, P., 1974. Heat transfer of Banks of Tubes in Crossflow at High Reynolds Numbers. *Heat Exchangers: Design and Theory Source Book*, by Afgan, N. and Schlünder, E.U., Scripta Book Company, Washington D.C., Mc Graw Hill, ISBN: 0-07-000460-9: 74-100.



CURRICULUM VITAE

Name : Azize AKKOCA

Date of birth : March 08, 1970

Place of birth : Tarsus- İÇEL

Nationality : Turkish

Present position: Research Assistant in the Mechanical Engineering Department,
Çukurova University, ADANA, TURKEY

Address : Çukurova University, Mechanical Engineering Department,
01330, Balcali, ADANA, TURKEY

Telephone : +90-322-3386084 (Ext: 2722)

Fax : +90-322-3386126

GSM : +90-0-533-6584770

E- mail : aakkoca@cu.edu.tr
azize@mersin.edu.tr

RESEARCH ACTIVITY

Fluid Flow and Heat Transfer Characteristics of Compact Heat Exchangers,
Computational Fluid Dynamics (CFD), Computational Heat Transfer (CHT),
FIDAP, PHOENICS, GAMBIT, and FLUENT CFD codes, PIV Technique.

EDUCATIONAL BACKGROUND

1982 : Tarsus Duatepe Primary School

1984 : Tarsus Kasım Ekenler High School

1987 : Tarsus Lycee

1994 : **BSc**, University of Çukurova, Mechanical Engineering Department

1998 : **MSc**, University of Mersin, Mechanical Engineering Department

1998-present : **PhD**, University of Çukurova, Mechanical Engineering Department

EMPLOYMENT

1995- 2000 :Research Assistant, Mersin University, Institute of Natural and Applied Sciences, MERSİN, TURKEY

2000-2002 :Research Assistant, Mersin University, Faculty of Engineering, MERSİN, TURKEY

2002-present :Research Assistant, Çukurova University, Institute of Natural and Applied Sciences, ADANA, TURKEY

MEMBERSHIP OF INSTITUTIONS

The American Society of Mechanical Engineers (ASME)

Union of Chambers of Turkish Engineers and Architects, Chamber of Mechanical Engineers (TMMOB Makina Mühendisleri Odası)

APPENDIX A

DERIVATION OF CONSERVATION EQUATIONS

There are many methods to derive the conservation equations. One of the simplest, a *control volume approach*, is used here to demonstrate the origin of each term. These equations may be used to analyze the flow of most common fluids in internal (pipes) or external (wings) flow situations.

Derivation of Continuity Equation

Let's start with a small, fixed volume of fluid somewhere in the middle of a flow stream. This elemental volume has sides of lengths Δx , Δy and Δz (Figure 1).

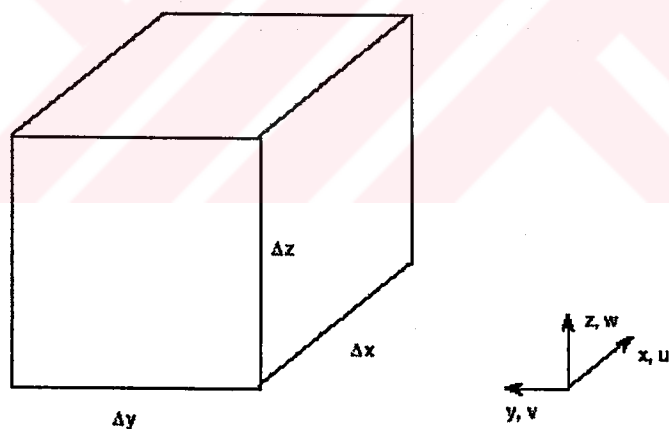


Figure 1. Illustration of the elemental volume used to derive the equations.

These lengths are short enough so that changes in all fluid properties across the volume may be well approximated with linear functions. On the other hand, these dimensions must be large enough so that the fluid may be considered as a continuum (i.e., much larger than the molecular scale). The mass of fluid in this elemental volume depends on the amount of fluid entering and leaving through the faces. The difference between these two is the rate of mass that accumulates in the volume. The

rate of mass entering a face is the product of the density, the fluid velocity and the face area. For example, on the side facing the reader, the density (ρ) is multiplied by the velocity in the x direction (u) and the area of the face $\Delta y \Delta z$. Thus, the mass flux entering the volume through this face is

$$\text{mass flux in} = \rho u \Delta y \Delta z$$

The mass leaving the volume on the opposite side of the volume is again the product of density, velocity and area, but the density and velocity may have changed as the fluid passed through the volume. We will express these changes as small quantities (since our volume is small enough), i.e., $\rho + \Delta\rho$ and $u + \Delta u$. The mass flux leaving that face is thus

$$\text{mass flux out} = -(\rho + \Delta\rho)(u + \Delta u) \Delta y \Delta z$$

The negative sign tells us that the mass is leaving, rather than entering, the control volume. Performing the same analysis on the mass entering the volume through the other faces of the volume gives us

$$\rho v \Delta x \Delta z$$

$$\rho w \Delta x \Delta y$$

where v is the y direction velocity and w is the z direction velocity. Similarly, the mass fluxes leaving the volume on the opposite faces are

$$-(\rho + \Delta\rho)(v + \Delta v) \Delta x \Delta z$$

$$-(\rho + \Delta\rho)(w + \Delta w) \Delta x \Delta y$$

All of these added together must equal the mass of fluid accumulating in the volume, $\Delta x \Delta y \Delta z$,

$$\left(\frac{\Delta\rho}{\Delta t}\right)\Delta x\Delta y\Delta z$$

Putting all of these together, we have

$$\begin{aligned}\left(\frac{\Delta\rho}{\Delta t}\right)\Delta x\Delta y\Delta z &= \rho u\Delta y\Delta z + \rho v\Delta x\Delta z + \rho w\Delta x\Delta y \\ &- (\rho + \Delta\rho)(u + \Delta u)\Delta y\Delta z - (\rho + \Delta\rho)(v + \Delta v)\Delta x\Delta z - (\rho + \Delta\rho)(w + \Delta w)\Delta x\Delta y\end{aligned}$$

Multiplying out the quantities in parentheses results in the cancellation of some terms and the appearance of higher-order terms such as $\Delta\rho\Delta u\Delta y\Delta z$. Since the quantities preceded by Δ are very small, products of these quantities will be extremely small, depending on the number of Δ terms included in the product. The terms with four of these will be much smaller than the terms with only three Δ terms. Thus, all higher order terms are neglected. This leaves

$$\left(\frac{\Delta\rho}{\Delta t}\right)\Delta x\Delta y\Delta z = -(u\Delta\rho + \rho\Delta u)\Delta y\Delta z - (\rho\Delta v + v\Delta\rho)\Delta x\Delta z - (\rho\Delta w + w\Delta\rho)\Delta x\Delta y$$

or

$$\left(\frac{\Delta\rho}{\Delta t}\right)\Delta x\Delta y\Delta z = -\Delta(\rho u)\Delta y\Delta z - \Delta(\rho v)\Delta x\Delta z - \Delta(\rho w)\Delta x\Delta y$$

which, when divided by $\Delta x\Delta y\Delta z$ and rearranged, yields

$$\frac{\Delta\rho}{\Delta t} + \frac{\Delta(\rho u)}{\Delta x} + \frac{\Delta(\rho v)}{\Delta y} + \frac{\Delta(\rho w)}{\Delta z} = 0$$

The application of basic calculus (taking the limit as Δt tends to 0) allows us to write this equation as

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

where the symbol $\partial/\partial t$, for example, is a "partial derivative" with respect to time. Partial derivatives are used when the function (velocity or density in this case) depends on several variables (3 position or spatial variables and time, in this case).

The *Continuity Equation* may be simplified for some common flow situations as follows. If the fluid may be treated as incompressible (as is the case with water or in low velocity air flows), the density will be constant. The Continuity Equation then becomes

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \text{ (incompressible fluid)}$$

In the case when the flow is steady (all time derivatives are zero), then

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

Note that in this equation, the density and velocities are still functions of the spatial coordinates x, y and z .

Derivation of Momentum (Navier Stokes) Equations

Again we start with a small, fixed volume of fluid somewhere in the middle of a flow stream with sides of lengths Δx , Δy and Δz (Figure 1). The law of the conservation of momentum states that the rate of change of momentum in the control volume must equal the net momentum flux into the control volume plus any external forces acting on the control volume. We will first deal with the momentum change and flux, then with the external forces. Recall that momentum, the product of mass and velocity, is a vector quantity. This derivation will be based on the momentum in the x direction in Cartesian coordinates. Similar derivations may be demonstrated for the y and z direction. This would make a good exercise to better understand this material. There are thus three different momentum equations that together comprise the Navier-Stokes Equations.

Momentum Change and Flux

The time rate of change of momentum within the control volume is

$$\frac{\partial}{\partial t}(\rho u) \Delta x \Delta y \Delta z$$

where $\partial/\partial t$ is the partial derivative operator with respect to time presented above. The flux of momentum in the x direction into each face of the control volume is the product of the mass flux (mf x ΔA) and the x direction velocity,

$$mf \times u \Delta A$$

where ΔA is the surface area. For the side facing the reader, the momentum flux is thus

$$\rho u u \Delta y \Delta z$$

The momentum flux out the opposite side is

$$-\left(\rho u u + \frac{\partial}{\partial x}(\rho u u) \Delta x\right) \Delta y \Delta z$$

The mass flux into the face with normal vector in the negative y direction is as derived for the continuity equation above, or ρv . The momentum flux into this face is thus

$$\rho v u \Delta x \Delta z$$

For the face opposite, the momentum flux out is

$$-\left(\rho vu + \frac{\partial}{\partial y}(\rho vu)\Delta y\right)\Delta x\Delta z$$

Using a similar analysis, it is easily shown that the momentum flux into and out of the faces with normal vectors in the z direction are

$$\begin{aligned} &\rho wu\Delta x\Delta y \\ &\text{and} \\ &-\left(\rho wu + \frac{\partial}{\partial z}(\rho wu)\Delta z\right)\Delta x\Delta y \end{aligned}$$

Our first expression of the momentum equation comes from adding all of these terms together as expressed in the law of the conservation of momentum

$$\begin{aligned} &\frac{\partial(\rho u)}{\partial t}\Delta x\Delta y\Delta z = \rho uu\Delta y\Delta z - \left(\rho uu + \frac{\partial}{\partial x}(\rho uu)\Delta x\right)\Delta y\Delta z \\ &+ \rho vu\Delta x\Delta z - \left(\rho vu + \frac{\partial}{\partial y}(\rho vu)\Delta y\right)\Delta x\Delta z + \rho wu\Delta x\Delta y - \left(\rho wu + \frac{\partial}{\partial z}(\rho wu)\Delta z\right)\Delta x\Delta y + \sum F_x \end{aligned}$$

where $\sum F_x$ is the sum of the external forces on the control volume. The momentum fluxes into the control volume cancel with the first part of the momentum fluxes out of the control volume. Performing this cancellation and moving the momentum fluxes to the left hand side of the equation gives

$$\left(\frac{\partial(\rho u)}{\partial t} + \frac{\partial}{\partial x}(\rho uu) + \frac{\partial}{\partial y}(\rho vu) + \frac{\partial}{\partial z}(\rho wu)\right)\Delta x\Delta y\Delta z = \sum F_x$$

Using the product rule, the momentum change and fluxes can be expanded to

$$\left(\rho \frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} + \rho w \frac{\partial u}{\partial z} + u \frac{\partial \rho}{\partial t} + u \frac{\partial(\rho u)}{\partial x} + u \frac{\partial(\rho v)}{\partial y} + u \frac{\partial(\rho w)}{\partial z}\right)\Delta x\Delta y\Delta z = \sum F_x$$

where it is noted that the last four terms in parentheses are the continuity equation times u . Since this must be zero, that leaves

$$\left(\rho \frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} + \rho w \frac{\partial u}{\partial z} \right) \Delta x \Delta y \Delta z = \sum F_x$$

In the same method we can calculate the force in the y and z directions:

$$\left(\rho \frac{\partial v}{\partial t} + \rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y} + \rho w \frac{\partial v}{\partial z} \right) \Delta x \Delta y \Delta z = \sum F_y$$

$$\left(\rho \frac{\partial w}{\partial t} + \rho u \frac{\partial w}{\partial x} + \rho v \frac{\partial w}{\partial y} + \rho w \frac{\partial w}{\partial z} \right) \Delta x \Delta y \Delta z = \sum F_z$$

Derivation of Energy Equation

In situations where the fluid may be treated as incompressible and temperature differences are small, the continuity and momentum equations are sufficient to specify the velocities and pressure (that is, four equations [continuity+3 momentum] and four unknown quantities [u, v, w and p]). If the flow is compressible (ρ is not constant), or if heat flux occurs (temperature not constant), at least one additional equation is required. In some of these instances, the energy equation may be used. In the derivation, we use the fact that work is the dot product of velocity and force and the fact that work and energy are related to each other.

The energy equation is, of course, a scalar equation, meaning that it has no particular direction associated with it. The procedure for deriving the energy equation is similar to those presented for the continuity and momentum equations. In this case, the change in energy of the fluid within the control volume is equal to the net thermal energy transferred *into* the control volume plus the rate of work done by external forces. The energy of the fluid is expressed in this case as the sum of the absolute thermodynamic internal energy per unit mass, e , and the kinetic energy per

unit mass, $1/2 V^2$, (V is the magnitude of the velocity vector). The change in total energy per unit volume of the fluid in the control volume is

$$\frac{\partial}{\partial t} \left[\rho \left(e + \frac{1}{2} v^2 \right) \right]$$

As was found above for the momentum transfer into and out of the control volume, the net transfer of energy per unit volume through the control volume is

$$\frac{\partial}{\partial x} \left[\rho u \left(e + \frac{1}{2} v^2 \right) \right] + \frac{\partial}{\partial y} \left[\rho v \left(e + \frac{1}{2} v^2 \right) \right] + \frac{\partial}{\partial z} \left[\rho w \left(e + \frac{1}{2} v^2 \right) \right]$$

This equation is obtained by replacing the momentum term (density times velocity) by the energy term (density times the sum of the internal and kinetic energies). The net thermal energy transferred into the control volume is determined by the heat flux q_i , positive for heat going from within the control volume to the surroundings in the i^{th} -direction (that is, the x-, y- or z-direction). The total heat per unit volume transferred *into* the control volume is

$$-\frac{\partial q_x}{\partial x} - \frac{\partial q_y}{\partial y} - \frac{\partial q_z}{\partial z}$$

The rate of work per unit volume being done by the surface forces is found by multiplying the stress, σ_{ij} , by the velocity in the j -direction for each i face. Similar to the procedure above for the stresses in the momentum equation, the net rate of work being done from all sides is

$$\frac{\partial}{\partial x} (u \sigma_{xx} + v \sigma_{xy} + w \sigma_{xz}) + \frac{\partial}{\partial y} (u \sigma_{yx} + v \sigma_{yy} + w \sigma_{yz}) + \frac{\partial}{\partial z} (u \sigma_{zx} + v \sigma_{zy} + w \sigma_{zz})$$

Lastly, the rate of work per unit volume done by the gravity force vector ($\mathbf{g} = g_x \mathbf{i} + g_y \mathbf{j} + g_z \mathbf{k}$), is

$$\rho U g_x + \rho V g_y + \rho W g_z$$

Putting all of these terms together, we have

$$\begin{aligned} & \frac{\partial}{\partial t} \left[\rho \left(e + \frac{1}{2} v^2 \right) \right] + \frac{\partial}{\partial x} \left[\rho u \left(e + \frac{1}{2} v^2 \right) \right] + \frac{\partial}{\partial y} \left[\rho v \left(e + \frac{1}{2} v^2 \right) \right] + \frac{\partial}{\partial z} \left[\rho w \left(e + \frac{1}{2} v^2 \right) \right] = \\ & - \frac{\partial q_x}{\partial x} - \frac{\partial q_y}{\partial y} - \frac{\partial q_z}{\partial z} + \frac{\partial}{\partial x} (u \sigma_{xx} + v \sigma_{xy} + w \sigma_{xz}) + \frac{\partial}{\partial y} (u \sigma_{yx} + v \sigma_{yy} + w \sigma_{yz}) + \frac{\partial}{\partial z} (u \sigma_{zx} + v \sigma_{zy} + w \sigma_{zz}) \\ & + \rho U g_x + \rho V g_y + \rho W g_z \end{aligned}$$

Fourier's Heat Conduction

We will now use Fourier's Law of Heat Conduction that relates the heat flow in the i^{th} direction, q_i , to the rate of change of temperature in the i^{th} direction, namely,

$$q_i = -k_i A \partial T / \partial x_i \quad i=1,2,3 \text{ represents the } x,y,z \text{ directions}$$

where k_i is the heat conduction coefficient in the i^{th} -direction, A represents the surface area perpendicular to the i^{th} -direction, and T represents the temperature of the flow. From the Zeroth Law of Thermodynamics, heat flows from a location of higher temperature to that of a lower temperature. If, for example, this is in the x -direction, then $\partial T / \partial x$ is negative. But since heat flow is considered positive when flowing from the control volume to the surroundings (meaning, in this case, in the positive x -direction), then for q_x to be positive, we need the minus sign as indicated. Thus the heat flow rate per unit volume terms

$$- \frac{\partial q_x}{\partial x} - \frac{\partial q_y}{\partial y} - \frac{\partial q_z}{\partial z}$$

become

$$\partial (k_x \partial T / \partial x) / \partial x + \partial (k_y \partial T / \partial y) / \partial y + \partial (k_z \partial T / \partial z) / \partial z$$

which, in the case of constant heat conduction coefficient ($k_x = k_y = k_z = k$), become

$$k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right)$$

The rate of work per unit volume being done by the surface forces is found by multiplying the stress σ_{ij} by the velocity in the j direction for each i face. Similar to the procedure above for the stresses in the momentum equation, the net rate of work per unit volume being done from all sides is

$$\frac{\partial}{\partial x} (\mu \sigma_{xx} + v \sigma_{xy} + w \sigma_{xz}) + \frac{\partial}{\partial y} (\mu \sigma_{yx} + v \sigma_{yy} + w \sigma_{yz}) + \frac{\partial}{\partial z} (\mu \sigma_{zx} + v \sigma_{zy} + w \sigma_{zz})$$

Using the relationship between surface stresses and velocity gradients for incompressible Newtonian fluids, this becomes

$$\begin{aligned} & - \left(u \frac{\partial p}{\partial x} + v \frac{\partial p}{\partial y} + w \frac{\partial p}{\partial z} \right) \\ & + \mu \left[u \frac{\partial^2 u}{\partial x^2} + \frac{\partial}{\partial x} \left(v \frac{\partial v}{\partial x} + w \frac{\partial w}{\partial x} \right) + v \frac{\partial^2 u}{\partial y^2} + \frac{\partial}{\partial y} \left(u \frac{\partial u}{\partial y} + w \frac{\partial w}{\partial y} \right) + w \frac{\partial^2 u}{\partial z^2} + \frac{\partial}{\partial z} \left(u \frac{\partial u}{\partial z} + v \frac{\partial v}{\partial z} \right) \right] \\ & + 2\mu \left[\left(\frac{\partial u}{\partial x} \right)^2 + \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} + \left(\frac{\partial v}{\partial y} \right)^2 + \frac{\partial v}{\partial z} \frac{\partial w}{\partial y} + \left(\frac{\partial w}{\partial z} \right)^2 + \frac{\partial w}{\partial x} \frac{\partial u}{\partial z} \right] \end{aligned}$$

Lastly, the rate of work done per unit volume by gravity is

$$\rho u g_x + \rho v g_y + \rho w g_z$$

Putting all of these terms together, we have

$$\begin{aligned}
& \frac{\partial}{\partial t} \left[\rho \left(e + \frac{1}{2} v^2 \right) \right] + \frac{\partial}{\partial x} \left[\rho u \left(e + \frac{1}{2} v^2 \right) \right] + \frac{\partial}{\partial y} \left[\rho v \left(e + \frac{1}{2} v^2 \right) \right] + \frac{\partial}{\partial z} \left[\rho w \left(e + \frac{1}{2} v^2 \right) \right] = \\
& \quad k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) - \left(u \frac{\partial p}{\partial x} + v \frac{\partial p}{\partial y} + w \frac{\partial p}{\partial z} \right) \\
& + \mu \left[u \frac{\partial^2 u}{\partial x^2} + \frac{\partial}{\partial x} \left(v \frac{\partial v}{\partial x} + w \frac{\partial w}{\partial x} \right) + v \frac{\partial^2 u}{\partial y^2} + \frac{\partial}{\partial y} \left(u \frac{\partial u}{\partial y} + w \frac{\partial w}{\partial y} \right) + w \frac{\partial^2 u}{\partial z^2} + \frac{\partial}{\partial z} \left(u \frac{\partial u}{\partial z} + v \frac{\partial v}{\partial z} \right) \right] \\
& + 2\mu \left[\left(\frac{\partial u}{\partial x} \right)^2 + \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} + \left(\frac{\partial v}{\partial y} \right)^2 + \frac{\partial v}{\partial z} \frac{\partial w}{\partial y} + \left(\frac{\partial w}{\partial z} \right)^2 + \frac{\partial w}{\partial x} \frac{\partial u}{\partial z} \right] + \rho u g_x + \rho v g_y + \rho w g_z
\end{aligned}$$

This equation demonstrates that, per unit volume, the change in energy of the fluid moving through a control volume is equal to the rate of heat transferred into the control volume plus the rate of work done by surface forces plus the rate of work done by gravity. This expression of the energy equation is valid for most applications. However, some specialized situations may require additional terms representing the contributions of other sources (electromagnetic forces, etc.).

APPENDIX B

DEVELOPMENT OF THE GOVERNING EQUATIONS AT A POINT FOR A TURBULENT TIME AVERAGED FLOW

Applying the continuity equation and the Navier-Stokes equations to turbulent flows requires:

- Considering dimensions that appear uniform for turbulent time-averaged variables
- Considering the time-dependent terms for flows that appear steady for turbulent time-averaged variables
- Including the statistics of the turbulence and including this information in the specified initial and boundary conditions

Turbulent Time-Averaged Variables

- Define

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

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

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

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

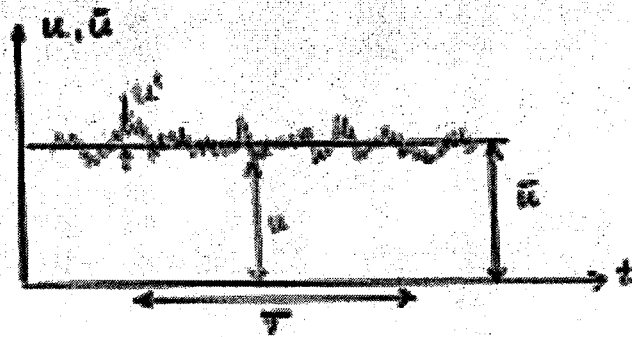
$$\rho = \bar{\rho} + \rho'$$

- Define the turbulent time-averaged variables as:

$$\bar{\alpha} = \frac{1}{T} \int_0^T \alpha dt$$

$$\alpha = u, v, w, p, \rho$$

T = an appropriate time period



- T is large compared to the time scale of the turbulent fluctuations.
- T is small compared to the time scale of variation of the turbulent time averaged flow.
- We assume that the behavior is ergodic.(i.e. the statistics are stationary for period T).
- Note that turbulence dissipates at the Komolgorov dissipation scale where the Reynolds number is

Re- smallest scale ≈ 1

- At this scale mechanical energy changes to heat. The large scale energy supply controls the smallest scale.

Turbulent Time-Averaged Form of the Continuity Equation

- Consider the Continuity Equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

- Substitute for the scale separated velocities

$$\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} + \frac{\partial w'}{\partial z} + \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0$$

- Time averaging the continuity equation we have:

$$\frac{1}{T} \int_0^T \left[\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} + \frac{\partial w'}{\partial z} + \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} \right] dt = 0$$

- Thus

$$\begin{aligned} & \frac{\partial}{\partial x} \left\{ \frac{1}{T} \int_0^T u' dt \right\} + \frac{\partial}{\partial y} \left\{ \frac{1}{T} \int_0^T v' dt \right\} + \frac{\partial}{\partial z} \left\{ \frac{1}{T} \int_0^T w' dt \right\} + \frac{\partial}{\partial x} \left\{ \frac{1}{T} \int_0^T \bar{u} dt \right\} \\ & + \frac{\partial}{\partial y} \left\{ \frac{1}{T} \int_0^T \bar{v} dt \right\} + \frac{\partial}{\partial z} \left\{ \frac{1}{T} \int_0^T \bar{w} dt \right\} = 0 \end{aligned}$$

- However, since $\bar{u}$ is determined as a constant over time T

$$\frac{1}{T} \int_0^T \bar{u} dt = \bar{u} \frac{1}{T} \int_0^T dt = \bar{u}$$

- Furthermore

$$\bar{u} = \frac{1}{T} \int_0^T u dt = \frac{1}{T} \int_0^T (\bar{u} + u') dt$$

$$\bar{u} = \frac{1}{T} \int_0^T \bar{u} dt = \frac{1}{T} \int_0^T (u') dt$$

$$\bar{u} = \frac{1}{T} \int_0^T \bar{u} dt = \frac{1}{T} \int_0^T (u') dt$$

- Thus

$$\frac{1}{T} \int_0^T u' dt = 0$$

- Hence, the previous equation simplifies to:

$$\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0$$

- However

$$\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} + \frac{\partial w'}{\partial z} + \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0$$

- Thus

$$\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} + \frac{\partial w'}{\partial z} = 0$$

- Thus the continuity equation applies to the molecularly averaged variables u, v, w ; to the turbulent time averaged variables $\bar{u}, \bar{v}, \bar{w}$; and to the turbulent fluctuations u', v', w' .
- In vector notation, the turbulent time averaged continuity equation is:

$$\nabla \cdot \bar{\mathbf{U}} = 0 \text{ where } \bar{\mathbf{U}} = \bar{u}\hat{i} + \bar{v}\hat{j} + \bar{w}\hat{k}$$

Turbulent Time Averaged Form of the Navier-Stokes Equations

- Consider the x -direction Navier-Stokes equation and substituting the expanded variables

$$\begin{aligned} & \frac{\partial(\bar{u} + u')}{\partial t} + (\bar{u} + u') \frac{\partial(\bar{u} + u')}{\partial x} + (\bar{v} + v') \frac{\partial(\bar{u} + u')}{\partial y} + (\bar{w} + w') \frac{\partial(\bar{u} + u')}{\partial z} \\ &= -\frac{1}{\rho_0} \frac{\partial(\bar{p} + p')}{\partial x} + \nu \frac{\partial^2(\bar{u} + u')}{\partial x^2} + \nu \frac{\partial^2(\bar{u} + u')}{\partial y^2} + \nu \frac{\partial^2(\bar{u} + u')}{\partial z^2} \end{aligned}$$

- Expanding and time averaging the entire equation over period T

$$\begin{aligned} & \overline{\frac{\partial \bar{u}}{\partial t}} + \overline{\frac{\partial u'}{\partial t}} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{u} \frac{\partial u'}{\partial x} + u' \frac{\partial \bar{u}}{\partial x} + u' \frac{\partial u'}{\partial x} \\ & + \bar{v} \frac{\partial \bar{u}}{\partial y} + \bar{v} \frac{\partial u'}{\partial y} + v' \frac{\partial \bar{u}}{\partial y} + v' \frac{\partial u'}{\partial y} \\ & + \bar{w} \frac{\partial \bar{u}}{\partial z} + \bar{w} \frac{\partial u'}{\partial z} + w' \frac{\partial \bar{u}}{\partial z} + w' \frac{\partial u'}{\partial z} \\ &= -\frac{1}{\rho_0} \frac{\partial \bar{p}}{\partial x} - \frac{1}{\rho_0} \frac{\partial p'}{\partial x} + \nu \frac{\partial^2 \bar{u}}{\partial x^2} + \nu \frac{\partial^2 u'}{\partial x^2} + \nu \frac{\partial^2 \bar{u}}{\partial y^2} + \nu \frac{\partial^2 u'}{\partial y^2} + \nu \frac{\partial^2 \bar{u}}{\partial z^2} + \nu \frac{\partial^2 u'}{\partial z^2} \end{aligned}$$

- Typical time averagings of products of time averaged variable terms:

$$\overline{\bar{u} \frac{\partial \bar{u}}{\partial x}} = \frac{1}{T} \int_0^T \bar{u} \frac{\partial \bar{u}}{\partial x} dt$$

$$\overline{u' \frac{\partial \bar{u}}{\partial x}} = \frac{1}{T} \bar{u} \frac{\partial \bar{u}}{\partial x} \int_0^T dt$$

$$\overline{\bar{u} \frac{\partial u'}{\partial x}} = \bar{u} \frac{\partial \bar{u}}{\partial x}$$

- Typical time averagings of products of time averaged terms and turbulent fluctuating components:

$$\overline{\bar{u} \frac{\partial u'}{\partial x}} = \frac{1}{T} \int_0^T \bar{u} \frac{\partial u'}{\partial x} dt$$

$$\overline{\bar{u} \frac{\partial u'}{\partial x}} = \frac{1}{T} \bar{u} \frac{\partial}{\partial x} \left[\int_0^T u' dt \right]$$

$$\overline{\bar{u} \frac{\partial u'}{\partial x}} = 0$$

- Time averagings of products of turbulent fluctuating components can not be simplified:

$$\overline{u' \frac{\partial u'}{\partial x}} = \frac{1}{T} \int_0^T u' \frac{\partial u'}{\partial x} dt$$

- This simplifies the time averaged x-direction Navier-Stokes equation to:

$$\begin{aligned} \frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial x} + \bar{w} \frac{\partial \bar{u}}{\partial x} = & -\frac{1}{\rho_0} \frac{\partial \bar{p}}{\partial x} + \nu \left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} + \frac{\partial^2 \bar{u}}{\partial z^2} \right) \\ & - \frac{1}{T} \int_0^T \left(u' \frac{\partial u'}{\partial x} + v' \frac{\partial u'}{\partial y} + w' \frac{\partial u'}{\partial z} \right) dt \end{aligned}$$

- We now add the continuity equation of the fluctuating components multiplied by within the time averaging integral (we are adding zero to the integral):

$$\begin{aligned} \frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} + \bar{w} \frac{\partial \bar{u}}{\partial z} = & -\frac{1}{\rho_0} \frac{\partial \bar{p}}{\partial x} + \nu \left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} + \frac{\partial^2 \bar{u}}{\partial z^2} \right) \\ & - \frac{1}{T} \int_0^T \left[\left(u' \frac{\partial u'}{\partial x} + v' \frac{\partial u'}{\partial y} + w' \frac{\partial u'}{\partial z} \right) + u' \left(\frac{\partial u'}{\partial x} + \frac{\partial u'}{\partial y} + \frac{\partial u'}{\partial z} \right) \right] dt \end{aligned}$$

- This leads to:

$$\begin{aligned} \frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} + \bar{w} \frac{\partial \bar{u}}{\partial z} = & -\frac{1}{\rho_0} \frac{\partial \bar{p}}{\partial x} + \nu \left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} + \frac{\partial^2 \bar{u}}{\partial z^2} \right) \\ & - \frac{1}{T} \int_0^T \left[u' \frac{\partial u'}{\partial x} + u' \frac{\partial u'}{\partial x} + v' \frac{\partial u'}{\partial y} + u' \frac{\partial v'}{\partial y} + w' \frac{\partial u'}{\partial z} + u' \frac{\partial w'}{\partial z} \right] dt \end{aligned}$$

- Finally we arrive at the final x-direction equation:

$$\begin{aligned} \frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} + \bar{w} \frac{\partial \bar{u}}{\partial z} = & -\frac{1}{\rho_0} \frac{\partial \bar{p}}{\partial x} + \nu \left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} + \frac{\partial^2 \bar{u}}{\partial z^2} \right) \\ & - \left(\frac{\partial (\overline{u'u'})}{\partial x} + \frac{\partial (\overline{u'v'})}{\partial y} + \frac{\partial (\overline{u'w'})}{\partial z} \right) \end{aligned}$$

- Grouping the viscous terms and turbulent time averaged fluctuations:

$$\begin{aligned} \frac{\partial \bar{u}}{\partial t} + \bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} + \bar{w} \frac{\partial \bar{u}}{\partial z} = & -\frac{1}{\rho_0} \frac{\partial \bar{p}}{\partial x} + \frac{\partial}{\partial x} \left(\nu \frac{\partial \bar{u}}{\partial x} - \overline{u'v'} \right) + \frac{\partial}{\partial y} \left(\nu \frac{\partial \bar{u}}{\partial y} - \overline{u'v'} \right) \\ & + \frac{\partial}{\partial z} \left(\nu \frac{\partial \bar{u}}{\partial z} - \overline{u'v'} \right) \end{aligned}$$

- Similarly, the y-direction turbulent time averaged Navier-Stokes equation is:

$$\begin{aligned} \frac{\partial \bar{v}}{\partial t} + \bar{u} \frac{\partial \bar{v}}{\partial x} + \bar{v} \frac{\partial \bar{v}}{\partial y} + \bar{w} \frac{\partial \bar{v}}{\partial z} = & -\frac{1}{\rho_0} \frac{\partial \bar{p}}{\partial y} + \frac{\partial}{\partial x} \left(\nu \frac{\partial \bar{v}}{\partial x} - \overline{v'u'} \right) + \frac{\partial}{\partial y} \left(\nu \frac{\partial \bar{v}}{\partial y} - \overline{v'v'} \right) \\ & + \frac{\partial}{\partial z} \left(\nu \frac{\partial \bar{v}}{\partial z} - \overline{v'w'} \right) \end{aligned}$$

- Similarly, the z-direction turbulent time averaged Navier-Stokes equation is:

$$\frac{\partial \bar{w}}{\partial t} + \bar{u} \frac{\partial \bar{w}}{\partial x} + \bar{v} \frac{\partial \bar{w}}{\partial y} + \bar{w} \frac{\partial \bar{w}}{\partial z} = -\frac{\bar{\rho}g}{\rho_0} - \frac{1}{\rho_0} \frac{\partial \bar{p}}{\partial z} + \frac{\partial}{\partial x} \left(\nu \frac{\partial \bar{w}}{\partial x} - \overline{w'u'} \right) + \frac{\partial}{\partial y} \left(\nu \frac{\partial \bar{w}}{\partial y} - \overline{w'v'} \right) + \frac{\partial}{\partial z} \left(\nu \frac{\partial \bar{w}}{\partial z} - \overline{w'w'} \right)$$

- These equations are referred to as the **Reynolds Equations**
- They are very similar to the Navier-Stokes equations except that:
- Variables are turbulent time averaged quantities
- There are now convective inertia forces caused by turbulent fluctuations
- There are now two mechanisms of momentum transfer built into these equations:
- type terms are the **viscous stresses** and represent the averaged effect of molecular motions. These terms are necessary since we are not directly simulate momentum transfer via molecular level collisions.
- type terms are the **turbulent Reynolds stresses** and represent the averaged effect of momentum transfer due to turbulent fluctuations. These terms are necessary when using turbulent time averaged variables since we are not directly simulating momentum transfer via turbulent fluctuations.
- Note that the effective stress is now

$$\frac{\tau_{ij}}{\rho} = \nu \frac{\partial \bar{u}_i}{\partial x_j} - \overline{u'_j u'_i}$$

- The Reynolds Equations greatly simplify turbulent flow computations in terms of:
- Often allow a reduction of the number and complexity of the governing equations.
- Simplify the time and space variability of the variables, allowing us to focus on much larger spatial and temporal scales.
- This is very useful in numerical models.
- The Reynolds Equations include 6 new unknown terms:

$$\overline{u'u'}, \overline{u'v'}, \overline{u'w'}, \overline{v'v'}, \overline{v'w'}, \overline{w'w'}$$

We can easily show that

$$\overline{u_i' u_j'} = u_j' u_i'$$

- We must introduce additional constitutive relationships to solve for these unknowns.
- While the constitutive relationships for molecular level averaging was fluid and fluid condition (temperature and pressure) dependent, the constitutive relationships for turbulent flow are both **fluid and flow dependent**.
- The simplest **closure** models are similar to a Newtonian stress-flow relationship. These may work well for very specific problems.
- An example of a simple turbulence closure model is the Boussinesq model for turbulence:

$$\frac{\tau'_{ij}}{\rho} = -u_i' u_j' = \frac{\mu_t}{\rho} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right)$$

where μ_t is the turbulent viscosity which is a flow dependent quantity.

- In vector notation the Reynolds equations are:

$$\frac{D\bar{U}}{Dt} = \frac{\bar{\rho} g k}{\rho_0} - \frac{1}{\rho_0} \nabla \bar{p} + \frac{1}{\rho_0} \nabla \cdot T_{t/\infty}$$

$$\bar{U} = \bar{u} \hat{i} + \bar{v} \hat{j} + \bar{w} \hat{k}$$

$$T_{t/\infty} = \begin{bmatrix} \tau^{t/\infty}_{xx} & \tau^{t/\infty}_{xy} & \tau^{t/\infty}_{xz} \\ \tau^{t/\infty}_{yx} & \tau^{t/\infty}_{yy} & \tau^{t/\infty}_{yz} \\ \tau^{t/\infty}_{zx} & \tau^{t/\infty}_{zy} & \tau^{t/\infty}_{zz} \end{bmatrix}$$

APPENDIX C

```
#include "udf.h"
```

```
DEFINE_PROFILE(unsteady_velocity, thread, position)
```

```
{
    face_t f;

    begin_f_loop(f, thread)
    {
        real t = RP_Get_Real("flow-time");
        F_PROFILE(f, thread, position) = 2 + 0.2*sin(62.8*t);
    }
    end_f_loop(f, thread)
}
```

```
#include "udf.h"
```

```
#define c_s0 0.15
```

```
#define c_a 0.10
```

```
DEFINE_TURBULENT_VISCOSITY(user_mu_t, c, t)
```

```
{
    real mu_t, c_s, sigma;
    real length_sgs=0.;
    real rho = C_R(c,t);
    real volum = C_VOLUME(c,t);
    real local_deriv, conv_deriv, total_deriv, current_timestep;

    C_UDSI(c,t,0)=C_STRAIN_RATE_MAG(c,t);
    current_timestep=RP_Get_Real("physical-time-step");
    local_deriv=(C_UDSI(c,t,0)-C_UDSI_M1(c,t,0))/current_timestep;

    conv_deriv=C_U(c,t)*C_UDSI_G(c,t,0)[0]+C_V(c,t)*C_UDSI_G(c,t,0)[1]+C_W(c,t)
    )
    *C_UDSI_G(c,t,0)[2];
    total_deriv=local_deriv+conv_deriv;
    sigma=total_deriv/pow(C_STRAIN_RATE_MAG(c,t),1);

    c_s=c_s0*(1+c_a*sigma);
    length_sgs=c_s*pow(volum,1./3.);
    /*les_smag_const=c_s/
    mu_t=rho*length_sgs*length_sgs*C_STRAIN_RATE_MAG(c,t);

    return mu_t;
}
```