

**ÇUKUROVA UNIVERSITY
INSTITUTE OF NATURAL AND APPLIED SCIENCES**

MSc THESIS

Münire ÖZTÜRK

**NUMERICAL INVESTIGATION OF A NON-NEWTONIAN
FLUID FLOW THROUGH A CONTRACTION PIPE**

DEPARTMENT OF MECHANICAL ENGINEERING

ADANA,2012

ÇUKUROVA UNIVERSITY
INSTITUTE OF NATURAL AND APPLIED SCIENCES

**NUMERICAL INVESTIGATION OF A NON-NEWTONIAN FLUID FLOW
THROUGH A CONTRACTION PIPE**

Münire ÖZTÜRK

MSc THESIS

DEPARTMENT OF MECHANICAL ENGINEERING

We certify that the thesis titled above was reviewed and approved for the award of degree of the Master of Science by the board of jury on 01/02/2012.

.....
Assoc.Prof.Dr. Ahmet PINARBAŞI
SUPERVISOR

.....
Prof.Dr. Hüseyin AKILLI
MEMBER

.....
Asst.Prof.Dr. Mehmet KARAKILÇIK
MEMBER

This MSc Thesis is written at the Department of Institute of Natural And Applied Sciences of Çukurova University.

Registration Number:

Prof. Dr. İlhami YEĞİNGİL
Director
Institute of Natural and Applied Sciences

Not:The usage of the presented specific declarations, tables, figures, and photographs either in this thesis or in any other reference without citation is subject to "The law of Arts and Intellectual Products" number of 5846 of Turkish Republic

ABSTRACT

MSc THESIS

NUMERICAL INVESTIGATION OF A NON-NEWTONIAN FLUID FLOW THROUGH A CONTRACTION PIPE

Münire ÖZTÜRK

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

Supervisor : Assoc. Prof. Dr. Ahmet PINARBAŞI

Year: 2012, Pages: 99

Jury : Assoc. Prof. Dr. Ahmet PINARBAŞI

: Prof. Dr. Hüseyin AKILLI

: Asst. Prof. Dr. Mehmet KARAKILÇIK

Flow characteristics near a sudden contraction for a pipe is analyzed in this study. The study consists of three phases. In the first phase, existing experimental results for water using a Particle Image Velocimetry (PIV) method performed by one of our group members were compared with numerical simulation results. In the second phase, for certain Re values, numerical simulation results for water and polyacrylamide solution, which is an inelastic non-Newtonian fluid, were compared. In the last phase, numerical simulations were performed for polyacrylamide considering smaller Reynolds numbers. The results of first phase studies show that experimental and numerical results agree with each other fairly well; demonstrating that the numerical software we use is correct and dependable. The results of second and third phase studies show that the differences between flow characteristics of Newtonian and non-Newtonian fluids. For the same Re, streamwise and transverse velocity values for non-Newtonian fluid are much smaller than the Newtonian fluid while the vorticity values for non-Newtonian fluid are higher than Newtonian fluid.

Key Words : Non-Newtonian Fluids, entry flow, sudden contractions

ÖZ

YÜKSEK LİSANS TEZİ

DARALAN KESİTLERDE NEWTONYEN OLMAYAN AKIŞKANIN AKIŞININ SAYISAL OLARAK İNCELENMESİ

Münire ÖZTÜRK

ÇUKUROVA ÜNİVERSİTESİ
FEN BİLİMLERİ ENSTİTÜSÜ
MAKİNA MÜHENDİSDİĞİ ANABİLİM DALI

Danışman : Doç. Dr. Ahmet PINARBAŞI

Yıl: 2012, Sayfa: 99

Jüri : Doç. Dr. Ahmet PINARBAŞI

: Prof. Dr. Hüseyin AKILLI

: Yrd. Doç. Dr. Mehmet KARAKILÇIK

Bu çalışmada boru içerisindeki ani kesit daralması yakınlarında akış özellikleri analiz edilmiştir. Çalışma üç aşamadan oluşmaktadır. İlk aşamada, bölüm üyelerimizden birisi tarafından daha önce su ile PIV (Particle Image Velocimetry) yöntemi kullanılarak yapılmış olan deneysel çalışma, sayısal simülasyon sonuçları ile karşılaştırılmıştır. İkinci aşamada, belirli Re değerleri için su ve inelastik non-Newtonyen bir akışkan olan polyacrylamide çözeltisi ile yapılan sayısal simülasyonlar karşılaştırılmıştır. Son aşamada, polyacrylamide için daha küçük Reynolds sayılarını da gözlemlemek amaçlı değerlerle simülasyonlar yapılmıştır. İlk aşamadaki sonuçlar deneysel ve sayısal simülasyon sonuçlarının birbirini oldukça iyi bir şekilde doğruladığını gösterdi, bu da kullanılan yazılımın doğru ve güvenilir olduğunu göstermektedir. Çalışmanın ikinci ve üçüncü aşamasındaki sonuçlar, Newtonyen ve non-Newtonyen akışkanların akış özellikleri arasındaki farklılıkları göstermektedir. Aynı Reynolds sayısı için, non-Newtonyen akışkan için elde edilen akış yönündeki ve akışın enine yöndeki hız değerleri Newtonyen akışkana göre çok daha küçük iken, non-Newtonyen akışkan için elde edilen vorteks değerlerinin Newtonyen akışkana göre daha yüksek olduğu görülmüştür.

Anahtar Kelimeler: Newtonyen olmayan akışkan, daralan kesitte akış

ACKNOWLEDGEMENTS

This dissertation could not have been written without the help and support of Assoc. Prof. Dr. Ahmet PINARBAŞI who not only served as my supervisor but also encouraged and challenged me throughout my academic program. I am heartily thankful to him for never accepting less than my best efforts.

Also, I want to thank Prof. Dr. Hüseyin AKILLI for always sharing his ideas with me about my study and guidance from the initial to the final level of my thesis.

I would like to thank Research assistant Bengi GÖZMEN, Erhan FIRAT and Mehmet UÇAR for all helps, moral support and excellent friendship.

I want to thank Research assistant Murat YAVUZ for all helps and excellent friendship.

Finally I would also like to thank my mother and my father and especially my husband Hakan DEMİRKIRAN for their interest and help.

CONTENTS	PAGE
ABSTRACT	I
ÖZ	II
ACKNOWLEDGEMENTS	III
CONTENTS	IV
LIST OF FIGURES	VI
LIST OF TABLES.....	X
NOMENCLATURE	XII
1. INTRODUCTION	1
2. PREVIOUS STUDIES.....	7
2.1. Flow Analysis of a Newtonian Fluid.....	7
2.2.Flow Analysis of Inelastic Non-Newtonian Fluid.....	9
2.3. Flow Analysis of Viscoelastic Non-Newtonian Fluid.....	10
2.4. Flow Analysis of Polymer Solutions.....	11
2.5. Computational Numerical Analysis.....	12
3. MATERIAL AND METHOD.....	15
3.1. Introduction	15
3.2. Non-Newtonian Fluids	15
3.2.1. Time-independent fluid behaviour	16
3.2.1.1. Shear-thinning or pseudoplastic fluids	17
3.2.1.2 Viscoplastic fluid.....	18
3.2.1.3 Shear-thickening or dilatant fluid	19
3.2.2. Polymeric Liquids	19
3.2.2.1 Polyacrylamide	20
3.3. Finite Element Method	22
3.4. Ansys FLUENT Numerical Simulation Software	23
3.4.1. Pre-Processing Step	24
3.4.2. Simulation Step	25
3.4.3. Post-Processing Step	26
4. RESULTS AND DISCUSSIONS	27

4.1 Comparison of PIV Method and Numerical Simulation Results for Water.....	27
4.1.1. Comparison of Velocity Vectors, Streamlines and Contours	29
4.1.2. Comparison of Axial and Radial Velocity Profiles	54
4.2 Comparison of Water and Polyacrylamide Numerical Solutions Results	65
4.3 Results of Numerical Simulation for aqueous polyacrylamide solution.....	81
5. CONCLUSION	91
5.1 Recommendations for Future Studies	92
REFERENCES	93
CURRICULUM VITAE	99

LIST OF FIGURES

PAGE

Figure 1.1. Schematic view of axisymmetric sudden contraction geometry	2
Figure 1.2. Schematic view of experimental Setup	5
Figure 3.1. Flow curves for time-independent fluids shear stress against shear rate..	17
Figure 3.2. Flow curves for time-independent fluids apparent viscosity against shear rate	17
Figure 3.3. Schematic view of test section	24
Figure 3.4. Pre-Processing Steps of Numerical Solutions with Ansys FLUENT	25
Figure 3.5. Post-Processing Steps of Numerical Solutions with Ansys FLUENT ...	26
Figure 4.1. Average velocity vectors at three different location for Re=233.	29
Figure 4.2. Streamlines for Re=233	30
Figure 4.3. Average u velocity contour for Re=233.....	31
Figure 4.4. Average v velocity contour for Re=233.....	32
Figure 4.5. Average w vorticity contour for Re=233.	33
Figure 4.6. Average velocity vectors at three different location for Re=336	34
Figure 4.7. Streamlines for Re=336	35
Figure 4.8. Average u velocity contour for Re=336.....	36
Figure 4.9. Average v velocity contour for Re=336.....	37
Figure 4.10. Average w vorticity contour for Re=336.	38
Figure 4.11. Average velocity vectors at three different location for Re=883	39
Figure 4.12. Streamlines for Re=883.....	40
Figure 4.13. Average u velocity contour for Re=883.....	41
Figure 4.14. Average v velocity contour for Re=883.....	42
Figure 4.15. Average w vorticity contour for Re=883.	43
Figure 4.16. Average velocity vectors at three different location for Re=1325	44
Figure 4.17. Streamlines for Re=1325.....	45
Figure 4.18. Average u velocity contour for Re=1325.....	46
Figure 4.19. Average v velocity contour for Re=1325.....	47
Figure 4.20. Average w vorticity contour for Re=1325.	48
Figure 4.21. Average velocity vectors at three different location for Re=1989	49

Figure 4.22. Streamlines for $Re=1989$	50
Figure 4.23. Average u velocity contour for $Re=1989$	51
Figure 4.24. Average v velocity contour for $Re=1989$	52
Figure 4.25. Average w vorticity contour for $Re=1989$	53
Figure 4.26. Radial velocity profiles V_x at $Re= 233$	55
Figure 4.27. Axial velocity profiles V_y at $Re= 233$	56
Figure 4.28. Radial velocity profiles V_x at $Re= 336$	57
Figure 4.29. Axial velocity profiles V_y at $Re= 336$	58
Figure 4.30. Radial velocity profiles V_x at $Re= 883$	59
Figure 4.31. Axial velocity profiles V_y at $Re= 883$	60
Figure 4.32. Radial velocity profiles V_x at $Re= 1325$	61
Figure 4.33. Axial velocity profiles V_y at $Re= 1325$	62
Figure 4.34. Radial velocity profiles V_x at $Re= 1989$	63
Figure 4.35. Axial velocity profiles V_y at $Re= 1989$	64
Figure 4.36. Average velocity vectors of numerical solution at three different location (0.5D, 1.17D, 1.75D) for $Re=233$	67
Figure 4.37. Streamlines of numerical simulations for $Re=233$	68
Figure 4.38. Average u velocity contour of numerical solutions for $Re=233$	69
Figure 4.39. Average v velocity contour of numerical solutions for $Re=233$	70
Figure 4.40. Average w vorticity contour of numerical solutions for $Re=233$	71
Figure 4.41. Average V_x velocity profiles for different y/D ratios of numerical solutions at $Re=233$	72
Figure 4.42. Average V_y velocity profiles for different y/D ratios of numerical solutions at $Re=233$	73
Figure 4.43. Average velocity vectors of numerical solution at three different location (0.5D, 1.17D, 1.75D) for $Re=336$	74
Figure 4.44. Streamlines of numerical simulations for $Re=336$	75
Figure 4.45. Average u velocity contour of numerical solutions for $Re=336$	76
Figure 4.46. Average v velocity contour of numerical solutions for $Re=336$	77
Figure 4.47. Average w vorticity contour of numerical solutions for $Re=336$	78

Figure 4.48. Average V_x velocity profiles for different y/D ratios of numerical solutions at $Re=336$	79
Figure 4.49. Average V_y velocity profiles for different y/D ratios of numerical solutions at $Re=336$	80
Figure 4.50. Velocity vectors and streamlines of numerical simulation with Polyflow for $Re= 1$	82
Figure 4.51. Velocity contours and vorticity contours of numerical simulation with Polyflow for $Re= 1$	83
Figure 4.52. Average V_x and V_y profiles of numerical simulation with Polyflow for $Re= 1$	84
Figure 4.53. Velocity vectors and streamlines of numerical simulation with Polyflow for $Re= 10$	85
Figure 4.54. Velocity contours and vorticity contours of numerical simulation with Polyflow for $Re= 10$	86
Figure 4.55. Average V_x and V_y profiles of numerical simulation with Polyflow for $Re= 10$	87
Figure 4.56. Velocity vectors and streamlines of numerical simulation with Polyflow for $Re= 100$	88
Figure 4.57. Velocity contours and vorticity contours of numerical simulation with Polyflow for $Re= 100$	89
Figure 4.58. Average V_x and V_y profiles of numerical simulation with Polyflow for $Re= 100$	90

LIST OF TABLES	PAGE
Table 3.1. Values of the material parameters for the test samples.....	22

NOMENCLATURE

ΔP_{EN}	: Entry pressure drop
τ_w	: Shear stress at wall
C'	: Loss coefficient
n'_c	: Couette correction
N'_{Re}	: Generalized Reynolds number
n	: Power Law index
γ	: Shear rate
η_0	: Viscosity at zero shear rate
λ	: Time constant
D_d	: Downstream tube diameter
D_u	: Upstream tube diameter
L_e	: Entry length
β	: Contraction ratio

1. INTRODUCTION

The study of entry-flow problems, in which a fluid flowing through a duct of large cross-section progresses through a contraction into a smaller one, has been the subject of numerous studies with Newtonian and Non-Newtonian fluids (Sousaa, P.C. et al., 2009). Such flows are commonly referred to as entry flows. Polymeric fluids are classified as non-Newtonian fluid and their flow behavior is generally more complex than their Newtonian counterparts. The flow of Non Newtonian fluids in channels is very important subject of engineering applications such as injection molding and extrusion processes. Flow behavior of polymeric liquids in polymer processing affects both the quality of products and process efficiency (Yamamoto, T. et al., 2003). Elastic instabilities in such flows may limit the processing speed and may often result in a defective product (Oztekin, A. et al., 2000). In polymer processing with complicated shapes, the flow in the process is essentially three-dimensional; therefore, it becomes important to understand three-dimensional flow behavior of polymeric liquids. Nowadays computer analyzing and simulation technology is capable of solving this kind of problems. This technology is useful for industry due to the time and high cost savings that caused by the trial and error or some other methods. The increasing use of computational tools to design industrial equipment for polymer processing requires the ability of existing numerical methods to predict accurately the flow of viscoelastic fluids (Alves, M. A. et al., 2003).

The basic elements for laminar flow through an abrupt circular contraction are illustrated in Figure 1.1. The flow progresses from being fully developed at a plane some distance upstream from the contraction to being fully developed in the downstream tube at a distance L_e from the contraction plane. The entry length L_e is the distance from the contraction plane ($z = 0$) required for the centerline velocity to become 98 or 99% of its fully developed value. Depending on the Reynolds number of the flow and the fluid type, a secondary-flow vortex may be present in the corner of the upstream tube. The characteristics of this vortex are of particular interest in the design of extrusion dies.

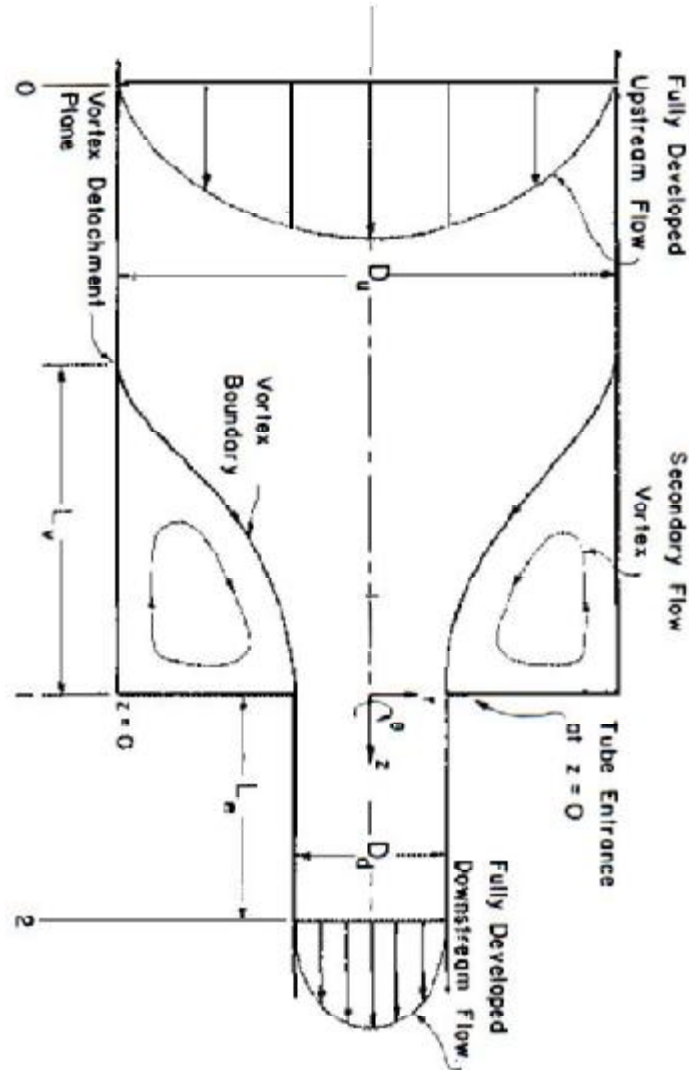


Figure 1.1. Schematic view of axisymmetric sudden contraction geometry. (Boger 1987)

As with other confined flows, the major objective is to predict the detailed kinematics and pressure drop using the equations of motion and an appropriate constitutive equation to define the stress tensor. Pressure drop for shear-thinning fluids in circular entry flow for $\beta \geq 4$ formulated by Boger (1987).

$$\frac{\Delta P_{EN}}{2\tau_W} = (C' + 1) \frac{N'_{Re}}{32} + n'_c \quad (1.1)$$

where C' loss coefficient and n'_c Couette correction,

Generalized Reynolds number

$$N'_{Re} = \frac{\rho D_d^{n'} V_d^{2-n'}}{8^{n'-1} K'} \quad (1.2)$$

is defined so that the Fanning function factor, $f = 16/N'_{Re}$, is valid for any inelastic fluid. Here n' is the slope of a log-log plot of τ_w versus $8V/D$ obtained with a capillary rheometer, while K' is the intercept of this plot at $8V/D = 1 \text{ S}^{-1}$. For a power-law fluid, we have $n = n'$ and

$$K = K' \left[\frac{4n}{3n+1} \right]^n \quad (1.3)$$

Computational Fluid Dynamics (CFD) is a science that, with the help of digital computers, produces quantitative predictions of fluid-flow phenomena based on those conservation laws (conservation of mass, momentum, and energy) governing fluid motion. These predictions normally occur under those conditions defined in terms of flow geometry, the physical properties of a fluid, and the boundary and initial conditions of a flow field. The prediction generally concerns sets of values of the flow variables, for example, velocity, pressure, or temperature at selected locations in the domain and for selected times. It may also evaluate the overall behavior of the flow, such as the flow rate or the hydrodynamic force acting on an object in the flow (Kundu, P.K., Cohen, I.M., 2008).

As a research and design tool, CFD normally complements experimental and theoretical fluid dynamics. However, CFD has a number of distinct advantages:

- Ø It can be produced inexpensively and quickly. Although the price of most items is increasing, computing costs are falling. According to Moore's law based on the observation of the data for the last 40 years, the CPU power will double every 18 months into the foreseeable future.

- Ø It generates complete information. CFD produces detailed and comprehensive information of all relevant variables throughout the domain of interest. This information can also be easily accessed.
- Ø It allows easy change of the parameters. CFD permits input parameters to be varied easily over wide ranges, thereby facilitating design optimization.
- Ø It has the ability to simulate realistic conditions. CFD can simulate flows directly under practical conditions, unlike experiments, where a small- or a large-scale model may be needed.
- Ø It has the ability to simulate ideal conditions. CFD provides the convenience of switching off certain terms in the governing equations, which allows one to focus attention on a few essential parameters and eliminate all irrelevant features.
- Ø It permits exploration of unnatural events. CFD allows events to be studied that every attempt is made to prevent, for example, conflagrations, explosions, or nuclear power plant failures (Kundu, P.K., Cohen, I.M., 2008).

In this thesis the previous experimental study of Fakılar, M.S. (2004), in which the working fluid is water, is simulated by Computational Fluid Dynamics (CFD) software and then the results are compared to show that simulation software properly works. Finally, Polyacrylamide solution in water, which is a non-Newtonian inelastic fluid, is simulated at different Reynolds numbers to be able to understand the flow behavior of polymeric fluids in a 4:1 constricted pipe.

The experimental test section of the study of Fakılar, M.S. (2004) is shown in Figure 1.2. We studied the same contraction geometry numerically using CFD to verify the efficiency of the commercially available CFD simulation program. The flow geometry studied is an axisymmetric sudden contraction consisting of two co-axially-connected circular tubes. The ratio of diameters of tubes, which is kept constant in this study, is defined as the contraction ratio $\beta = R_1/R_2 = 4$ where R_1 is 32 mm and R_2 is 8 mm. They show respectively, the radius of the upstream and downstream tubes. The lengths of the upstream (L_u) and downstream tube (L_d) are

respectively 96 R₂ and 24 R₂. The test section is made of Plexiglas (acrylic) to facilitate flow visualization. The fluid is forced through the test section from the first tank by free fall. The flow pressure is kept constant at atmospheric pressure. Another tank, which supplies water to the inlet tank, ensures a constant flow rate at the inlet tank. Also a by-pass pipe was used to keep constant water level. Pressure variations due to changes in the fluid level in the inlet tank are minimal. The length of the straight pipe upstream of the test section is long enough to provide fully developed flow conditions. This avoids the transport of any deformation history of the fluid into the test section due to upstream connections (elbow, valves, etc.).

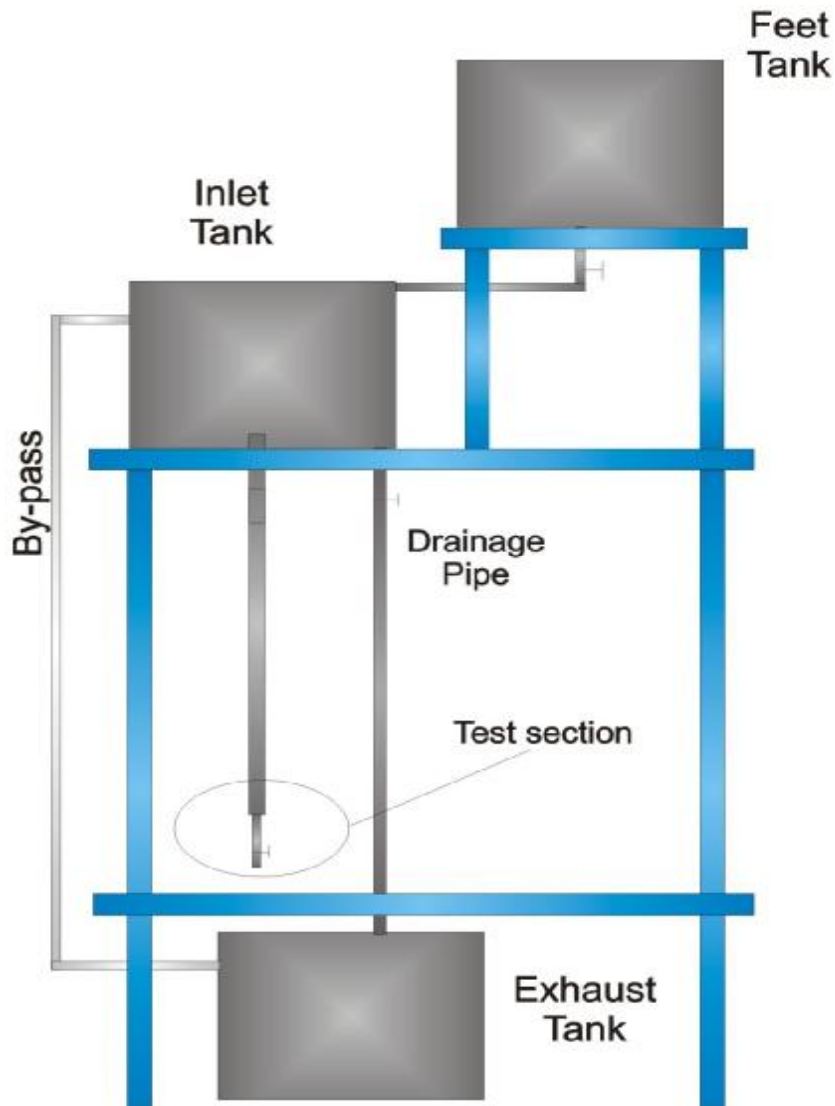


Figure 1.2. Schematic view of experimental setup (Fakılar, 2004)

2. PREVIOUS STUDIES

Interest in the entry-flow problem extends back to the late 1800s, when early workers like Hagenback (1860), Boussinesq (1891), and Couette (1890) were interested in the pressure drop for circular entry flows. Their investigations were motivated by a need to develop the capillary rheometer for accurate viscosity measurement of Newtonian fluids [see Kestin et al., (1973) for a review on the use of a capillary rheometer for viscosity measurement of Newtonian fluids].

After the publication of boundary-layer theory, there followed a great number of papers considering the flow of a Newtonian fluid from a flat entry velocity profile at plane 1 in Figure 1.1 to a fully developed flow at plane 2. Flow upstream of the tube entry was not considered. This is the so-called hydrodynamic entrance-region problem, where the main concern is predicting the entry length L_e . There was little interest in establishing whether the entry velocity profile was indeed flat under any condition. Most authors were more concerned with developing approximate techniques (integral and numerical) for solution of the boundary-layer equations than in establishing the detailed kinematics of entry flows. A complete bibliography on the hydrodynamic entrance region problem, published by Fan and Hwang (1966).

2.1. Flow Analysis of a Newtonian Fluid

Most of the studies of Newtonian entrance flow behaviour involve three major assumptions. First of all, excess viscous pressure dissipation in the reservoir is neglected. In the second place, the velocity profile is assumed to be uniform at the entrance of the smaller tube. Finally, it is postulated that the flow can be adequately described by replacing the complete, elliptic Navier-Stokes equations with their parabolic, asymptotic forms, the boundary-layer equations. These assumptions limit the results to high Reynolds number flows since axial diffusion of vorticity becomes increasingly important, as the Reynolds number is lowered presumably leading to considerable profile development and excess pressure dissipation upstream from the entrance of the smaller tube. Also, application of the boundary-layer equations is

effectively the result of formulating a singular perturbation problem around the large Reynolds number limit. Even at high Reynolds numbers it is not clear just how valid the first two assumptions are for entrances, which cause sharp changes in flow direction. It is also well known that the results of the boundary-layer analysis are not uniformly valid, yielding questionable values for the early stages of profile development in the smaller tube. Despite their shortcomings, analyses of the entrance flow problem utilizing the above three assumptions should, in some sense, provide a useful high Reynolds number asymptote to a more complete solution of the problem (Vrentas, et al., 1966).

Summaries of such entrance flow solutions and their relation to experimental data have been provided by Schmidt and Zeldin (1969). Weissberg (1962) considered flow from an infinitely large reservoir into a circular tube at the creeping flow limit, applying the minimum energy theorem of Helmholtz to arrive at an upper bound for the excess pressure drop through a sharp-edged entrance. The result of this study is a useful asymptote which complements the high Reynolds number limit, but, of course, the details of the flow field cannot be derived from such a variational method.

A realistic entrance model was utilized by Chu (1965) who examined the problem of flow Newtonian fluid through a sharp-edged entrance, but his results are quite limited.

Recent effort has been directed towards obtaining more general results by relaxing some of the above assumptions. Vrentas, Duda, and Barger (1966) numerically solved the complete equations of motion for entrance flow for Reynolds numbers ranging from 0 to 250 by replacing the upstream reservoir with an impermeable, frictionless, infinite tube of the same diameter as the circular conduit connected to the reservoir. Such a stream tube model is useful for examining the effect of axial diffusion of vorticity on velocity profile development, but it obviously oversimplifies the flow field in the reservoir. An analytical solution in the creeping flow limit provides some confirmation of the finite-difference result of the above study. Friedmann, Gillis, and Liron (1968) solved the complete Navier-Stokes equations for the Reynolds number range of 0-500 but retained the assumption of an

initially uniform velocity profile. Such a boundary condition is artificial for the case of low Reynolds numbers since there is no way to eliminate axial diffusion of vorticity at a particular point in the flow field unless special devices are inserted in the flow stream.

The results of Carter (1969) are more extensive, but some of his reported values of excess pressure drop are less than would be predicted by considering the kinetic energy change alone.

Christiansen et al., (1972) solved the general equations of motion numerically for the laminar isothermal flow of Newtonian fluids from a large tube of circular cross section through an abrupt contraction into a coaxial tube of smaller diameter and through the flow-development region of the smaller tube.

Vrentas and Duda (1973) solved the full Navier-Stokes equations by an explicit finite difference method for sudden contraction flow of a Newtonian fluid. Streamlines, vorticity distributions, velocity profiles, excess pressure drops and entrance lengths are calculated as functions of Reynolds number and contraction ratio.

2.2. Flow Analysis of Inelastic Non-Newtonian Fluid

Approximate boundary-layer techniques were first applied for the entry flow of inelastic non-Newtonian fluids (power-law fluids) by Bogue (1959), Collins and Schowalter (1963).

A comparison of the boundary-layer, creeping-flow, and earlier finite-difference solutions for the complete equations of motion with available experimental data for Newtonian and inelastic non-Newtonian fluids flowing through circular contractions is contained in a review published in 1982 (Boger 1982).

Fester et al. (2008) studied energy losses in the flows of both Newtonian and non-Newtonian fluids through sudden contractions of three contraction ratios $\beta=0.22, 0.5, 0.85$ with test fluids of water, glycerol solutions, lubrication oil, kaolin suspensions and carboxymethylcellulose (CMC) at Reynolds numbers ranged from approximately 0.01 to 10000.

Sousa et al. (2009) investigated the flow of a Newtonian fluid and a Boger fluid through sudden square-square contractions experimentally, aiming to characterize the flow and provide quantitative data for benchmarking in a complex three dimensional flow. Pressure drop measurements were performed in various geometries with different contraction ratios. For the Newtonian fluid, the experimental results are compared with numerical simulations performed and excellent agreement was found for the range of Reynolds numbers.

2.3. Flow Analysis of Viscoelastic Non-Newtonian Fluid

Kim-E et al., (1983) studied the roles of fluid inertia and shear-rate dependent viscosity in determining the velocity field in an axisymmetric sudden contraction with finite-element analysis for a generalized Newtonian fluid with viscosity function given by a Carreau equation.

Song and Yoo (1986) investigated the numerical simulation of the flow of upper convected Maxwell fluid through a planar 4:1 contraction using type dependent difference approximation of the vorticity equation. For creeping flow assumption, the numerical convergence has been achieved up to much higher values of the elasticity parameter than those obtained by conventional finite difference methods.

Yesilata et al., (1998) investigated the kinematics and dynamics of the viscoelastic flow in an axisymmetric 4:1 sudden contraction geometry for a highly elastic polyisobutylene (PIB) based polymer solution (referred to as PIB-Boger fluid). The critical conditions for the onset of the elastic instabilities and the dynamics of the resulting secondary flows are measured for various flow rates.

Yesilata et al., (1999) studied nonlinear dynamics of non-isothermal viscoelastic flow in an axisymmetric sudden contraction geometry for a highly elastic polyisobutylene (PIB) based polymer solution. The critical conditions for the onset of the elastic instabilities at different wall temperatures are determined by accurate instantaneous measurements of pressure drop across the contraction plane and temperature measurements in the flow.

Oztekin et al., (2000) investigated the viscoelastic flows of a PIB/ PB/ C14 polymer solution in axisymmetric cavity geometries. Multiple elastic flow transitions of spatial and temporal types are observed. The critical conditions for the onset of the instabilities and the spatial and temporal structures of the flow are examined using instantaneous differential pressure measurements and flow visualization by streakline photography.

Alves et al., (2004) investigated a numerical study of the creeping flow of a PTT fluid through planar sudden contractions carried out to quantify the effect of contraction ratio upon the flow characteristics (streamlines and size and intensity of recirculation vortices).

Oliveira et al., (2007) investigated a comprehensive numerical study of the effects of the contraction ratio upon viscoelastic flow through axisymmetric contractions. Six contraction ratios were examined (CR=2, 4, 10, 20, 40 and 100). The results are enabled the construction of vortex maps.

2.4. Flow Analysis of Polymer Solutions

Murthy and Boger (1971) studied the developing velocity profiles on the downstream side of a 2 to 1 contraction for inelastic polymer solutions. The profiles were measured using a technique involving streak photography. The test fluids employed were water and aqueous solutions of Methocel.

James and Saringer (1979) investigated experimentally and theoretically the behaviour of dilute polymer solutions in sink flow, viz., radial flow toward a point. Solutions of polyethylene oxide, in the drag-reducing concentration range, were pushed through a 60° conical channel at Reynolds numbers of order 10^2 .

Wunderlich et al., (1987) studied the flow of partially hydrolized polyacrylamide solutions through a 10:1.2 sudden planar contraction by means of laser-Dropper anemometry. The resultant velocity profiles are compared with those for Newtonian water flow.

2.5. Computational Numerical Analysis

Current major interest is the development of numerical methods in non-Newtonian fluid mechanics, where the circular entry flows is one of the problems being used to establish and test finite-element simulation techniques for viscoelastic fluids (Crochet and Walters 1983, Crochet et al. 1984).

As to viscoelastic fluids, the first published work in this area appears to be that performed by Mendelson et al., (1982). That work, however, was based on the use of a too simplistic viscoelastic fluid model, i.e. the so-called second-order model, rendering results not so consistent with experimental observations.

In the computational domain, early attempts dealing with the flow of non-Newtonian fluids in two and three-dimensional enclosures were concerned with purely-viscous non-Newtonian fluids. Reddy and Reddy (1992), for example, relied on the so-called power-law model to investigate heat transfer effects in a three-dimensional enclosure revealing the strong effects of a fluid's shear-thinning behavior on the flow kinematics.

Isaksson and Righdal (1994) studied the streamline pattern for the flow of a Carreau fluid in a slanted two-dimensional cavity in order to simulate flow in blade coating process.

In a more robust work, Grillet et al., (1999) relied on the FENE-Chillcot–Rallison model to study the effects of fluid elasticity on the flow kinematics and stress distribution with some success.

In a similar work, Tanguy et al., (2001) studied non-isothermal flow of a fluid obeying Carreau model in a square enclosure but failed to produce converged results even at low Reynolds numbers. Mitsoulis (2001) investigated lid-driven cavity flow of a viscoplastic fluid depicting the effect of yield stress on the flow inside the cavity.

To most researchers engaged in Computational Fluid Dynamics, the low Reynolds number flow of a highly viscous Newtonian fluid would not be considered as a challenging research topic. But add even a minute amount of macromolecules into the fluid, thus producing a polymer solution, and the situation is altered

drastically: the rheological (i.e. flow) behavior of the material becomes highly non-Newtonian, resulting in intricate flow phenomena whose prediction requires sophisticated modeling approaches and numerical tools.

3. MATERIAL AND METHOD

3.1. Introduction

Over the last three decades, possibilities for solving complex engineering problems have increased quite dramatically thanks to the advent of more powerful computers, more robust numerical methods, and more efficient algorithms. Currently, a wide range of computer-intensive problems in applied mechanics can be solved using well-established numerical methods such as Finite Difference Method (FDM), Finite Volume Method (FVM), Finite Element Method (FEM), and Boundary Element Method (BEM). The usage of computers has increased the popularity of computational fluid dynamic (CFD) analysis which is one of the analysis techniques, used in this thesis. Various CFD analysis package programs are available in commercial usage. Ansys FLUENT CFD Software is used in this thesis.

3.2. Non-Newtonian Fluids

A non-Newtonian fluid is one whose flow curve (shear stress versus shear rate) is nonlinear or does not pass through the origin, i.e. where the apparent viscosity, shear stress divided by shear rate, is not constant at a given temperature and pressure but is dependent on flow conditions such as flow geometry, shear rate, etc. and sometimes even on the kinematic history of the fluid element under consideration. Such materials may be conveniently grouped into three general classes:

(1) fluids for which the rate of shear at any point is determined only by the value of the shear stress at that point at that instant; these fluids are variously known as ‘ time independent ’ , ‘ purely viscous ’ , ‘ inelastic ’ or ‘ generalized Newtonian fluids ’ (GNF);

(2) more complex fluids for which the relation between shear stress and shear rate depends, in addition, upon the duration of shearing and their kinematic history; they are called ‘ time-dependent fluids ’ , and finally,

(3) substances exhibiting characteristics of both ideal fluids and elastic solids and showing partial elastic recovery, after deformation; these are categorized as ‘visco-elastic fluids’.

This classification scheme is arbitrary in that most real materials often exhibit a combination of two or even all three types of non-Newtonian features. Generally, it is, however, possible to identify the dominant non-Newtonian characteristic and to take this as the basis for the subsequent process calculations. Also, as mentioned earlier, it is convenient to define an apparent viscosity of these materials as the ratio of shear stress to shear rate, though the latter ratio is a function of the shear stress or shear rate and/or of time.

3.2.1. Time-independent fluid behaviour

In simple shear, the flow behaviour of this class of materials may be described by a constitutive relation of the form,

$$\gamma_{yx} = f(\tau_{yx}) \quad (3.1)$$

or its inverse form,

$$\tau_{yx} = f_1(\gamma_{yx}) \quad (3.2)$$

This equation implies that the value of γ_{yx} at any point within the sheared fluid is determined only by the current value of shear stress at that point or vice versa. Depending upon the form of the function in equation (3.1) or (3.2), these fluids may be further subdivided into three types:

- (a) shear-thinning or pseudoplastic
- (b) viscoplastic
- (c) shear-thickening or dilatant

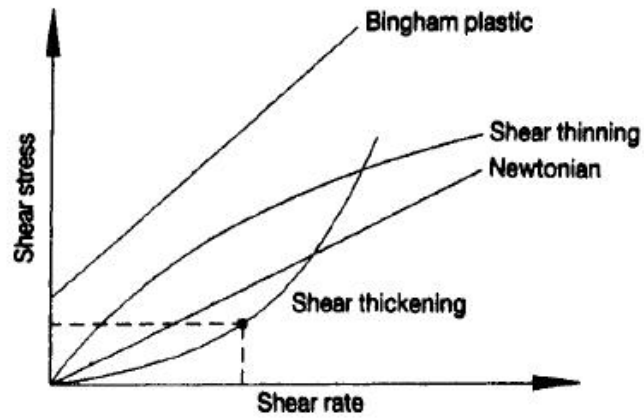


Figure 3.1. Flow curves for time-independent fluids. Shear stress against shear rate (Holland, 2002)

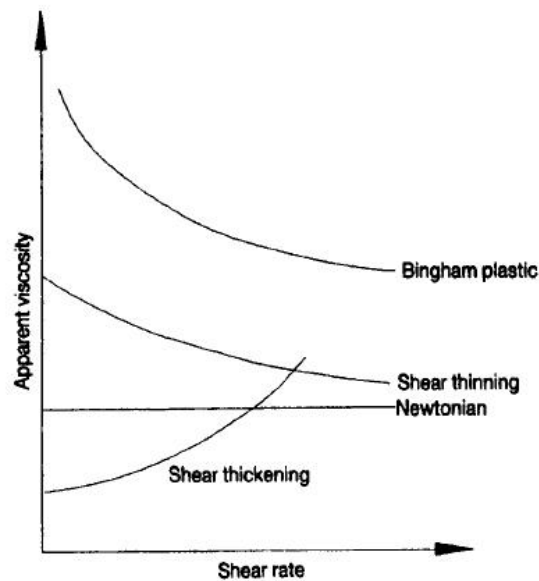


Figure 3.2. Flow curves for time-independent fluids. Apparent viscosity against shear rate (Holland, 2002)

3.2.1.1. Shear-thinning or pseudoplastic fluids

The most common type of time-independent non-Newtonian fluid behaviour observed is pseudoplasticity or shear-thinning, characterized by an apparent viscosity which decreases with increasing shear rate. Both at very low and at very high shear rates, most shear-thinning polymer solutions and melts exhibit Newtonian behaviour, i.e., shear stress–shear rate plots become straight lines, and on a linear scale will

pass through origin. The resulting values of the apparent viscosity at very low and high shear rates are known as the zero shear viscosity, μ_0 , and the infinite shear viscosity, μ_∞ , respectively. Thus, the apparent viscosity of a shear-thinning fluid decreases from μ_0 to μ_∞ with increasing shear rate.

Generally, the range of shear rate over which the apparent viscosity is constant (in the zero-shear region) increases as molecular weight of the polymer falls, as its molecular weight distribution becomes narrower, and as polymer concentration (in solution) drops. Similarly, the rate of decrease of apparent viscosity with shear rate also varies from one material to another.

3.2.1.2. Viscoplastic fluid

This type of fluid behavior is characterized by the existence of a yield stress (τ_0) which must be exceeded before the fluid will deform or flow. Conversely, such a material will deform elastically when the externally applied stress is smaller than the yield stress. Once the magnitude of the external stress has exceeded the value of the yield stress, the flow curve may be linear or non-linear but will not pass through origin.

A fluid with a linear flow curve for $|\tau_{yx}| > |\tau_0|$ is called a Bingham plastic fluid and is characterized by a constant plastic viscosity (the slope of the shear stress versus shear rate curve) and a yield stress. On the other hand, a substance possessing a yield stress as well as a non-linear flow curve on linear coordinates (for $|\tau_{yx}| > |\tau_0|$), is called a ‘ yield-pseudo plastic ’ material.

It is interesting to note that a viscoplastic material also displays an apparent viscosity which decreases with increasing shear rate. At very low shear rates, the apparent viscosity is effectively infinite at the instant immediately before the substance yields and begins to flow. It is thus possible to regard these materials as possessing a particular class of shear-thinning behavior

3.2.1.3. Shear-thickening or dilatant fluid

Dilatant fluids are similar to pseudoplastic systems in that they show no yield stress but their apparent viscosity increases with increasing shear rate; thus these fluids are also called shear-thickening. This type of fluid behaviour was originally observed in concentrated suspensions and a possible explanation for their dilatant behaviour is as follows: At rest, the voidage is minimum and the liquid present is sufficient to fill the void space. At low shear rates, the liquid lubricates the motion of each particle past others and the resulting stresses are consequently small. At high shear rates, on the other hand, the material expands or dilates slightly (as also observed in the transport of sand dunes) so that there is no longer sufficient liquid to fill the increased void space and prevent direct solid–solid contacts which result in increased friction and higher shear stresses.

3.2.2. Polymeric Liquids

Polymeric liquids are precisely what the name states: namely they are like the liquids we know that flow and have as constituents, some or all being long chain molecules or polymers. The conventional definition used to define simple liquids: that they do not support shear stress at rest, cannot be used to define the liquid state of a polymeric liquid. This is because polymeric liquids, like most other liquids are complex fluids: They exhibit both liquid and solid like behavior, and some of their dynamic properties may not be thermodynamic constants but some effective constant dependent on the history of forces acting on it. An example is a “viscosity” which is a function of shear rate, or a “viscosity” that changes with time.

The primary motivation to study polymeric fluids comes from industrial and commercial applications: Polymer processing and consumer products. These fields have provided (and continue to provide) with several problems, that are addressed by a wide variety of engineers and physicists, to the extent that has even earned a Nobel Prize in Physics for Pierre-Gilles de Gennes in 1991.

The development of a framework for theoretical prediction of the behavior of polymeric liquids in a flow has now occupied scientists, mathematicians, and engineers for at least the past fifty years. Today, it remains only partially resolved and is still the subject of intense investigation, spurred on by the development of new computational tools, new experimental techniques, and a major shift about twenty years ago in the way that we model and think about entangled polymer dynamics.

3.2.2.1. Polyacrylamide

The polyacrylamide solution is important for using coagulation of wastewater treatment process. Hence, the rheological behavior of polyacrylamide solution is very important in solution properties and engineering calculations related to the handling of the polymer solution for addition in wastewater treatment. The effect of various components of polyacrylamide solution on its flow behavior under low shear rate was recently reported. In the case of polyacrylamide solution, the rheological behavior of polyacrylamide homopolymer solution in water depends not only on shear rate, but also on variables such as concentration and temperature. The effect of various medium-concentration polyacrylamide solutions on its flow behavior was recently reported. It is unknown whether the flow behavior of polyacrylamide solution conforms to power law for a wide range of shear rates.

The following laws are available for shear-rate dependence of viscosity:

Ø Constant:

$$\eta(\dot{\gamma}) = \eta_0 \quad (3.3)$$

where η_0 is the viscosity at zero shear rate.

Ø Power Law:

The relationship between shear stress and shear rate for a shear-thinning fluid can often be approximated by a straight line over a limited range of shear rate (or stress).

$$F(\dot{\gamma}) = K * (\dot{\gamma})^{n-1} \quad (3.4)$$

for $n < 1$, the fluid exhibits shear-thinning properties

$n = 1$, the fluid shows Newtonian behaviour

$n > 1$, the fluid shows shear-thickening behaviour

Ø Bird-Carreau for shear thinning fluids:

This model can describe shear thinning behaviour over wide ranges of shear rates but only at the expense of the added complexity of four parameters.

$$F(\dot{\gamma}) = \eta_{\infty} + (\eta_0 - \eta_{\infty})(1 + \lambda^2 \dot{\gamma}^2)^{\frac{n-1}{2}} \quad (3.5)$$

Ø Bingham for yield stress materials:

This is the simplest equation describing the flow behaviour of a fluid with a yield stress and, in steady one-dimensional shear.

$$F(\dot{\gamma}) = \eta_0 + \frac{\tau_0}{\dot{\gamma}} \quad \dot{\gamma} \geq \dot{\gamma}_c \quad (3.6)$$

$$F(\dot{\gamma}) = \eta_0 + \frac{\tau_0 \left(2 - \frac{\dot{\gamma}}{\dot{\gamma}_c}\right)}{\dot{\gamma}_c} \quad \dot{\gamma} < \dot{\gamma}_c \quad (3.7)$$

Ø Cross:

This model reduces to the Newtonian fluid behaviour as $k \rightarrow 0$

$$\eta(\dot{\gamma}) = \frac{\eta_0}{1 + (\lambda \dot{\gamma})^m} \quad (3.8)$$

$$m = n - 1$$

Ø Carreau – Yasuda:

$$\eta(\dot{\gamma}) = \eta_{\infty} + (\eta_0 - \eta_{\infty})[1 + (\lambda\dot{\gamma})^b]^{\frac{n-1}{b}} \quad (3.9)$$

In this study, properties of the test liquid are taken from the experimental study of BROBOANĂ et al., (1987) in which properties of the test liquids have been obtained by adding to a Newtonian solvent (water with density $\rho_s = 1000 \text{ kg/m}^3$ and viscosity 1.05 mPas) pieces of concentrate solution of polyacrylamide with density $\rho_p = 1020 \text{ kg/m}^3$ (molecular weight $\text{MW} = 2 \cdot 10^5 \text{ g/mol}$) dissolved in water in different mass concentrations from C_{01} to C_{04} . The polymer solution with concentration C_i is obtained by mixing for one day the previous sample of concentration C_{i-1} with additional 2 grams of concentrate solution (the total mass of water used in experiments is 4 kg). shear experiments were performed with Physica MC 1 (double gap concentric cylinders) and Physica MC 300 (cone – plate geometry) was used for the oscillatory test. Material properties for different concentrations are given in Table 3.1. In this study, concentration C_{04} was adapted and used.

Table 3.1. Values of the material parameters for the test samples.

Material parameters	Concentration C_{01}	Concentration C_{02}	Concentration C_{03}	Concentration C_{04}
$\eta^* [\text{Pas}]$	0.0007	0.0016	0.0023	0.0028
$k [\text{s}]$	0.00009	0.00018	0.0002	0.00018
$n [-]$	0.0001	0.00137	0.1936	0.21

3.3. Finite Element Method

There are two ways for the formulation of the FEM: one is based on the direct variational method (such as the Rayleigh–Ritz method) and the other on the method of weighted residuals (such as the Galerkin method). In the formulation based on the variational method, the fundamental equations are derived from the stationary

conditions of the functional for the boundary-value problems. This formulation has an advantage that the process of deriving functionals is not necessary, so it is easy to formulate the FEM based on the method of the weighted residuals. In the formulation based on the variational method, however, it is generally difficult to derive the functional except for the case where the variational principles are already established as in the case of the principle of the virtual work or the principle of the minimum potential energy in the field of the solid mechanics. For most fluid dynamics problems, finite element applications have used the Galerkin finite element formulation.

3.4. Ansys FLUENT Numerical Simulation Software

There are huge numbers of engineering applications that can benefit from computational fluid dynamics simulation. By reducing the number of trial-and-error prototypes, simulation can save hundreds of thousands of dollars. In the polymer processing industry, cost savings can be a crucial consideration, with the price of tooling often part of the deciding factor in winning contracts.

Ansys FLUENT Software is a commercially available CFD program. Working principle of this software can basically define in three steps, pre-processing, solution and post-processing. Pre-processing step is also consists of two parts which are creating geometry and meshing. Because turbulence has a strong effect on flow characteristics over a wide range of applications FLUENT CFD Software offers a number of turbulence models. FLUENT Software can solve the problem correlated with some other Softwares at different stages at simulation such that GAMBIT can be used at Pre-Processing step for creating geometry, defining boundary conditions and meshing on the other hand TECHPLOT can be used at Post-Processing step to be able to get better visualization of results.

3.4.1. Pre-Processing Step

A closer view of test section, as well as dimensions and coordinate axis for the contraction problem is given below in Figure 3.3.



Figure 3.3. Drawing of test section

The main parameter in this study is effect of changing Reynolds number. This dimensionless parameter is defined for water as,

$$Re = \frac{\rho U_0 D}{\eta} \quad (3.10)$$

ρ : density of water, D : Diameter of upstream tube,

U_0 : inlet velocity, η : viscosity of water.

and for polyacrylamide as,

$$Re = \frac{\rho U_0 D}{\eta^*} \quad (3.11)$$

ρ : density of polyacrylamide and η^* : viscosity of polyacrylamide

In this study GAMBIT is used at Post-Processing step for creating geometry and meshing this geometry. Geometry of the problem is the one which is experimentally studied before by Fakilar, M.S., (2004). According to experimental apparatus, contraction ratio is defined as $\beta = D_1/D_2 = 4$ where D_1 is 64 mm and D_2 is 16 mm. Length of the upstream tube is $96 R_2$ which is equal to 768 mm and downstream tube is $24 R_2$ which is equal to 192 mm. After creating the geometry according to given values, created volume is meshed. Final step is defining the boundary conditions such as inlet area, wall, outlet area and exporting that geometry in file format of “.msh” to be able to import to FLUENT software.

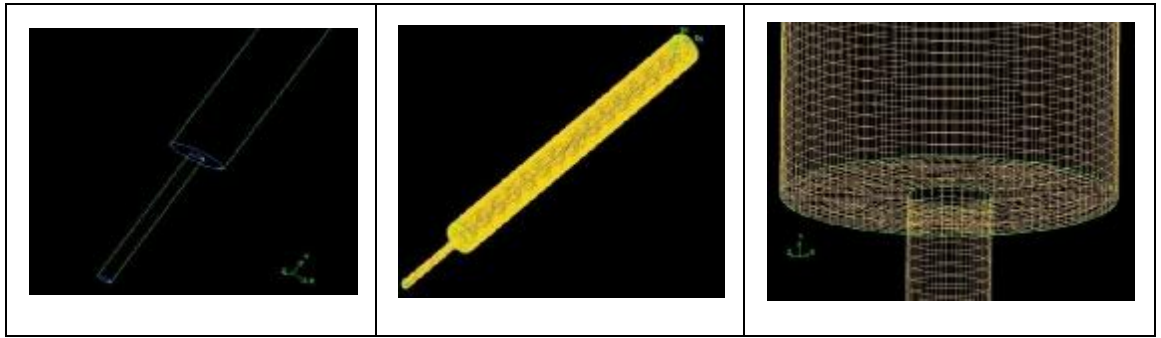


Figure 3.4. Pre-Processing Steps of Numerical Solutions with Ansys FLUENT

3.4.2. Simulation Step

At this step mesh file is imported to the FLUENT software then first thing is choosing the solution model such that the flow is laminar or turbulent and also if the flow is steady or unsteady and then material can be chosen if available in FLUENT library or we can define new material if it is not available. After defining operating condition as atmospheric condition, boundary conditions such as volumetric flowrate and pressure outlet for each Reynolds number is defined and then iteration is started for solution of 107211 nodes which we created by meshing the geometry. Iteration continues up to the converge value of 10^{-5} .

3.4.3. Post-Processing Step

After that the iteration is completed then results can be displayed by using display menu of FLUENT software. Also we can export the file in “.tech” format to be able to get better visual options by using TECHPLOT for displaying the results.

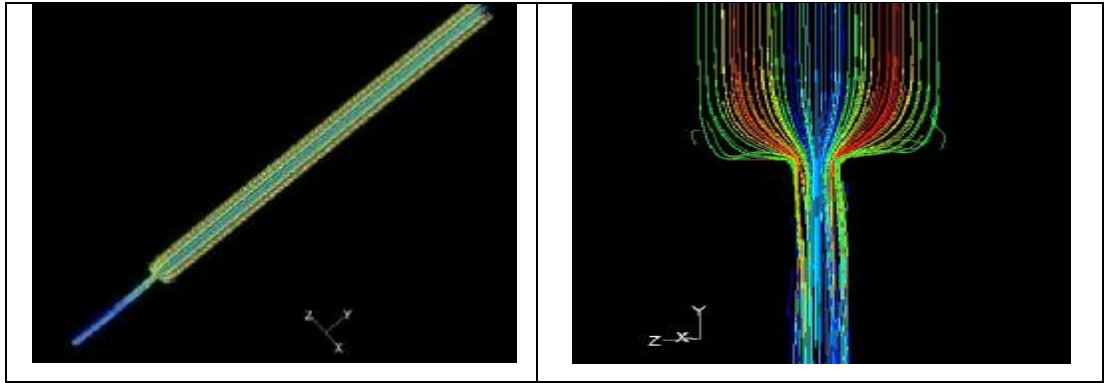


Figure 3.5. Post-Processing Steps of Numerical Solutions with Ansys FLUENT

Results are presented on plane which is $2D_d$ above the contraction line (where D_d is the diameter of downstream tube). Also lines are defined at $0.5 D_d$, $1.17 D_d$ and $1.75 D_d$ above the contraction plane for the purpose of displaying the velocity vectors same as in the previous study of Fakilar, M.S., (2004).

4. RESULTS AND DISCUSSIONS

4.1. Comparison of PIV Method and Numerical Simulation Results for Water

In the first phase of this study, water was chosen as working fluid which is readily available and defined at FLUENT Software database. Our goal here is to duplicate the study of Fakilar, M.S., (2004) to verify the effectiveness of CFD program we use. The observation window is defined between the orifice to $2D_d$ above the orifice where D_d is the diameter of downstream tube. The contraction ratio is defined as $\beta = D_u/D_d = 4$ for all numerical solutions where D_u is the upstream tube diameter. The obtained results of numerical solution are compared with the results of previous study which is done by PIV method to be able prove that software properly works. Numerical solutions are calculated for same Reynolds numbers as in previous study which are 233, 336, 883, 1325 and 1989. Velocity vectors are observed at different locations such as $0.5 D_d$, $1.17 D_d$ and $1.75 D_d$ above the orifice to be able to compare with previous study.

Figures 4.1, 4.6, 4.11, 4.16 and 4.21 show the comparison of average velocity vectors of experimental and numerical simulation results at $0.5D$, $1.17D$ and $1.75D$ above the orifice for Reynolds number values of 233, 336, 883, 1325 and 1989.

As can be seen from the Figures 4.1, 4.6, 4.11, 4.16 and 4.21, the directions of the velocity vectors are not affected about the middle of the upstream tube but the velocity vectors take higher values as Re increases. As the fluid approaches the orifice, the directions of the vectors are directed to the axis of contraction tube and the magnitudes of vectors are gets higher values. We can also see from the other Figures that as the Reynolds number increases the velocity vectors become larger and concentrate close together. We can also see that the Figure 4.1a and Figure 4.1b is similar to each other so the characteristics defined above is valid for both of them.

When we compare the figures of streamlines from Figures 4.2, 4.7, 4.12, 4.17 and 4.22 for increasing Reynolds numbers, we can clearly see that in numerical

simulation result a weak vortex occurs on the corners of upstream tube at $Re=233$, but as the Reynolds number increases, vortex disappears.

On the other Figures, we can see contours of average spanwise velocity u , average streamwise velocity v and corresponding vorticity w . Solid contour lines on these figures show positive velocities while dashed lines show negative velocity contours. It must be noted that the streamwise velocity v contours are always negative since the y axis is chosen positive upward and flow direction is downward. The value of velocities on the contours can be distinguished by colors which is shown on scales at the right side of figures. As can be seen from the figures highest value of spanwise velocity u , streamwise velocity v and the vorticity w is at the centre of contours. As can be seen from Figures 4.3, 4.8, 4.13, 4.18 and 4.23 when the Reynolds number increases, the value of spanwise velocity u increases and the contours are getting larger. Also from Figure 4.4, 4.9, 4.14, 4.19 and 4.24 values of streamwise velocity v increases due to the increase of Reynolds number so we can see that contours of v are getting larger. Finally from Figure 4.5 to 4.25 we can see the same effects of increasing Reynolds number on vorticity w .

Similarities in shapes of contours of PIV method and numerical solution we can see in the figures as well as similarities for velocity scale prove that the numerical solution software gives correct results.

Therefore, in the next section, the similar results for an inelastic fluid, namely the Polyacrylamide-water solution, will be given based on the simulations that were performed using Ansys software.

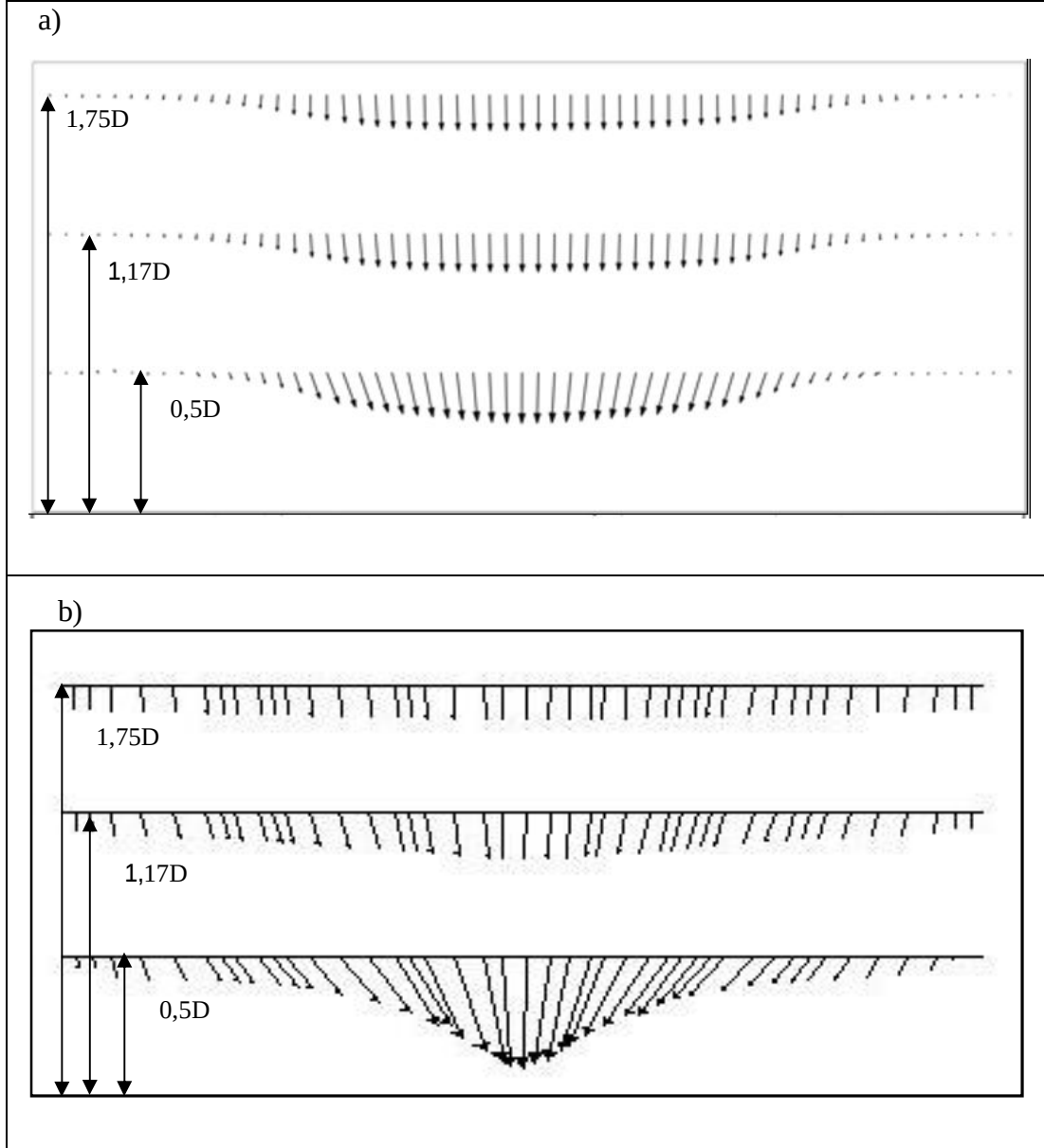
4.1.1. Comparison of Velocity Vectors, Streamlines and Contours

Figure 4.1. a) Average velocity vectors of PIV method at three different location for $Re=233$

b) Average velocity vectors of numerical solution at three different location for $Re=233$

From Figures 4.1a and 4.1b, we can see the similarities between the vectors at same distances above contraction plane. As fluid approaches the contraction plane

magnitudes of vectors gets higher and directions are directed to the centerline axis of tubes.

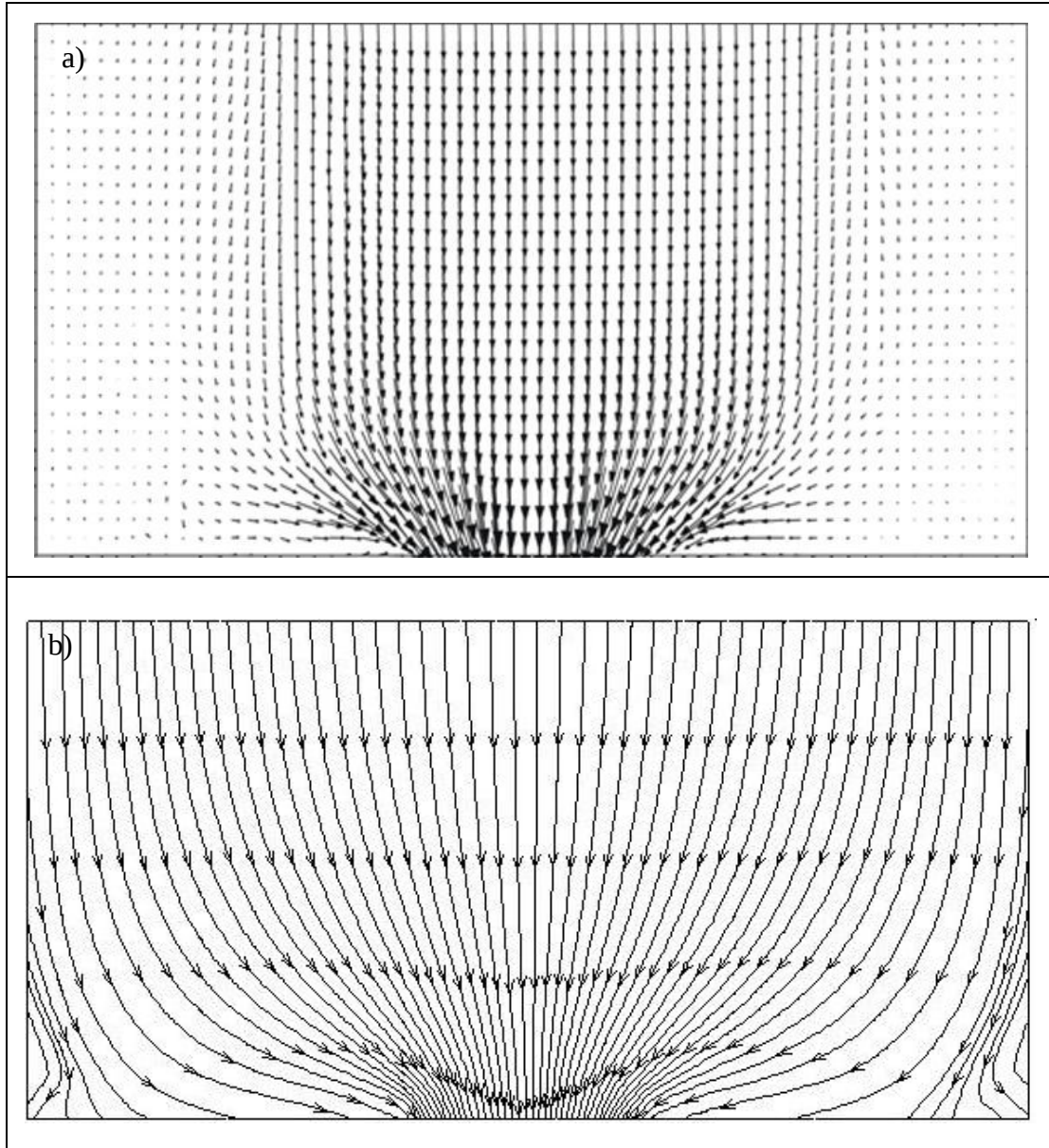


Figure 4.2. a) Streamline of experimental study for $Re=233$
b) Streamline of numerical simulation for $Re=233$

General streamline observations are similar, as can be seen from Figure 4.2b only difference is occurrence of vorticity for numerical simulation at Reynolds number of 233. Streamlines of experimental study in Figure 4.2a can not be seen clearly for what happens at the corners of upstream tube.

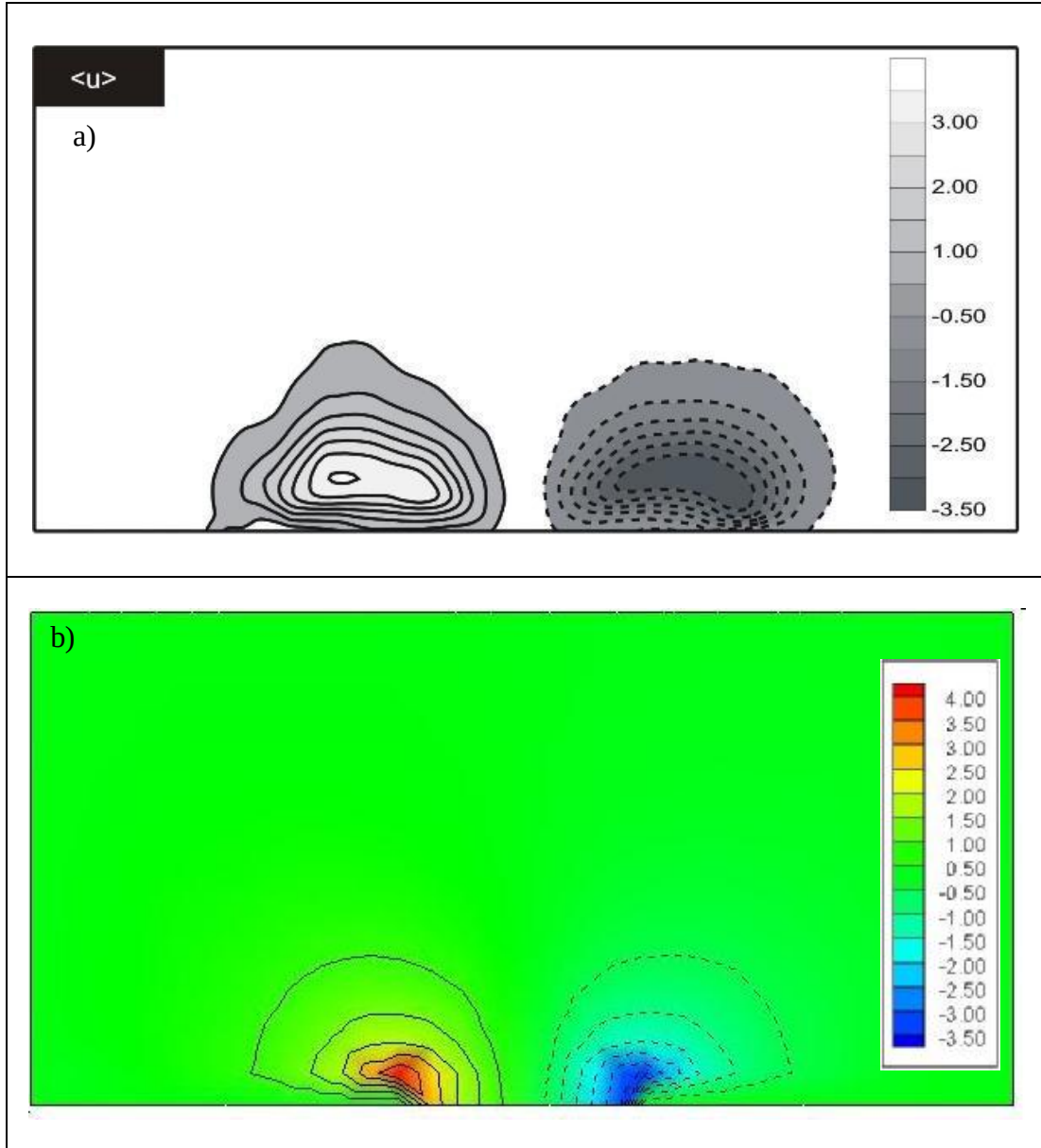


Figure 4.3. a) Average u velocity contour of PIV method for $Re=233$
 b) Average u velocity contour of numerical solution method for $Re=233$

When we compare the u velocity contours of experimental study and numerical simulation in Figures 4.3a and 4.3b, shapes of contours are very similar and also the velocity scales are the same. These similarities prove that the numerical simulation properly works.

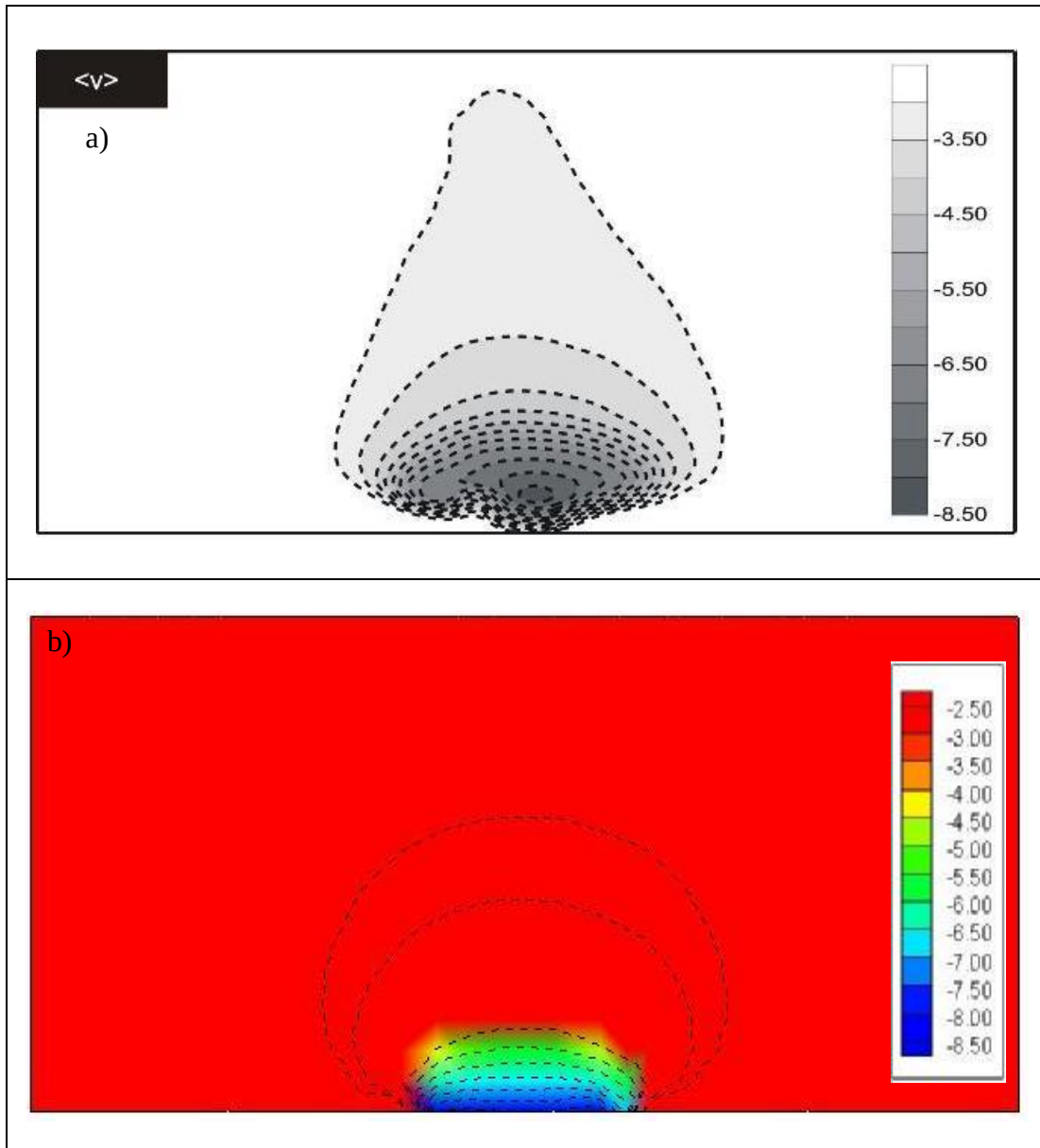


Figure 4.4. a) Average v velocity contour of PIV method for $Re=233$
 b) Average v velocity contour of numerical solution method for $Re=233$

Same as in contours of u velocity, we can see similarities between Figures 4.4a and 4.4b, in which the shape of the contours and scales are the same.

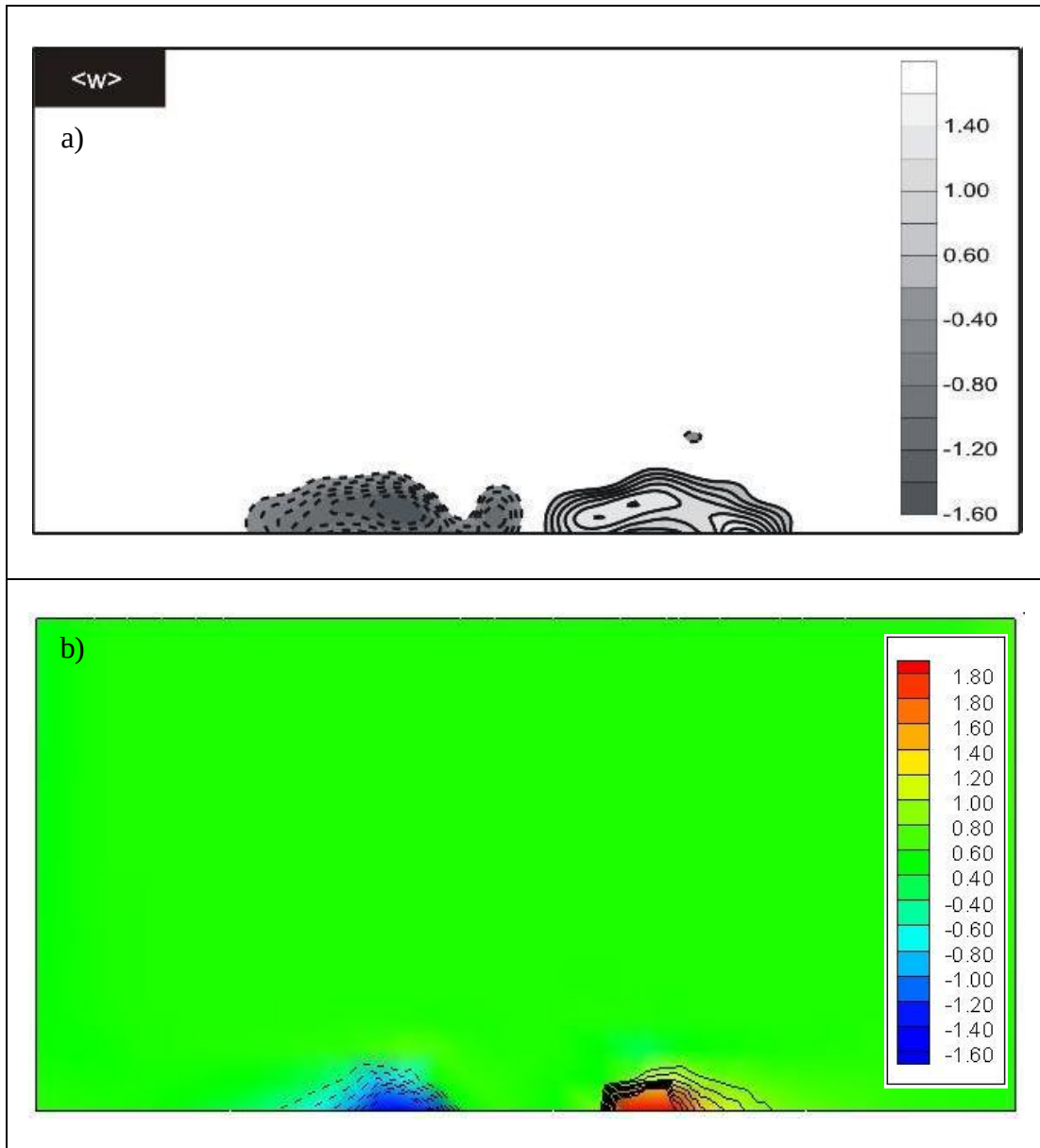


Figure 4.5. a) Average w vorticity contour of PIV method for $Re=233$

b) Average w vorticity contour of numerical solution method for $Re=233$

Experimental and numerical simulation results for w vorticity contours are the same, as can be seen from figures above. The difference is the distance between positive and negative contours.

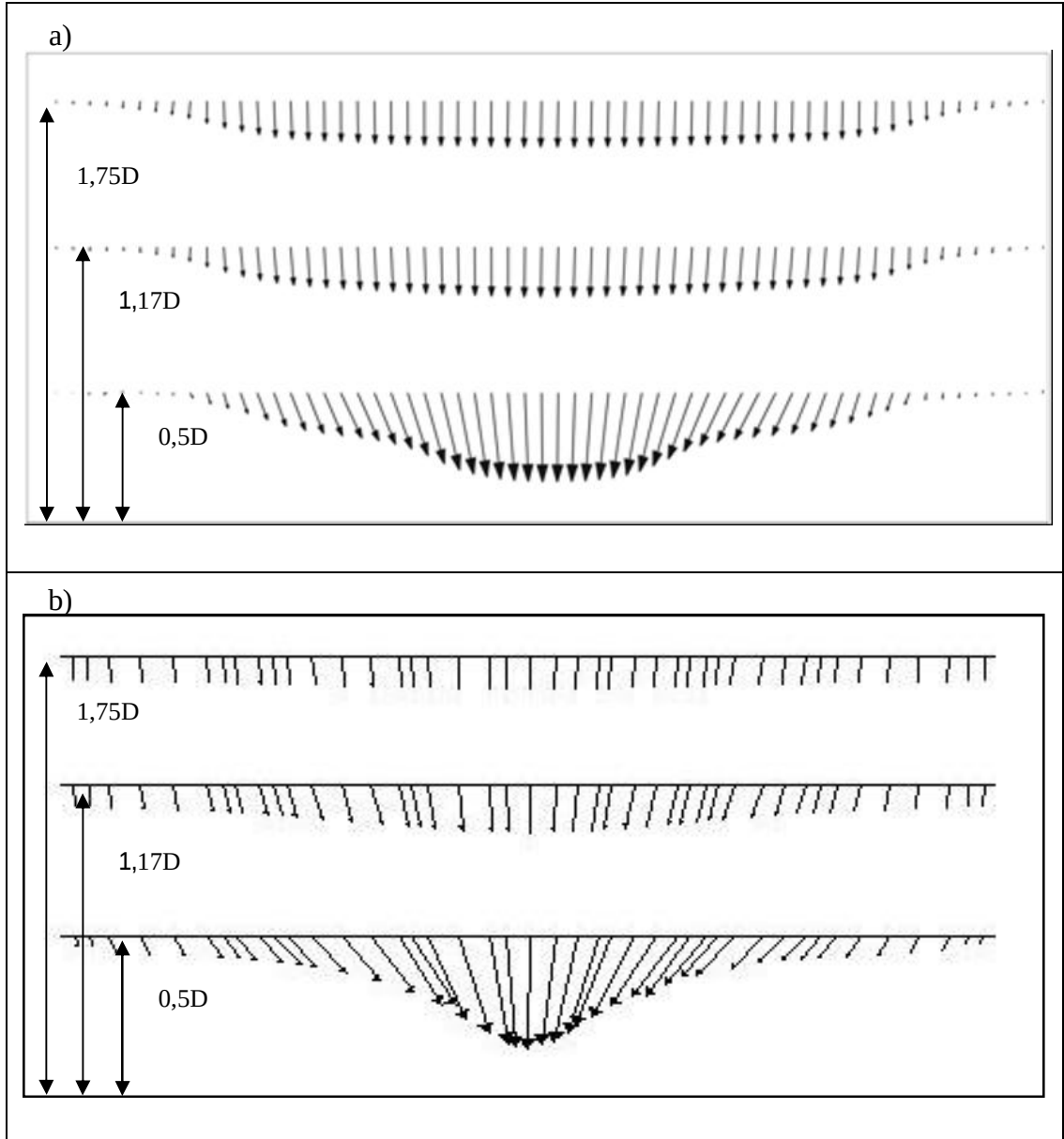


Figure 4.6. a) Average velocity vectors of PIV method at three different location for $Re=336$

b) Average velocity vectors of numerical solution at three different location for $Re=336$

As in Figures 4.1a and 4.1b, same similarities are observed in Figures 4.6a and 4.6b such that magnitudes of vectors increases as it approaches to contraction plane and vectors are directed towards to the centerline axis. Also when we compare the Figures 4.1 and 4.6, we can see that as the Reynolds number increases the magnitudes of vectors increases too.

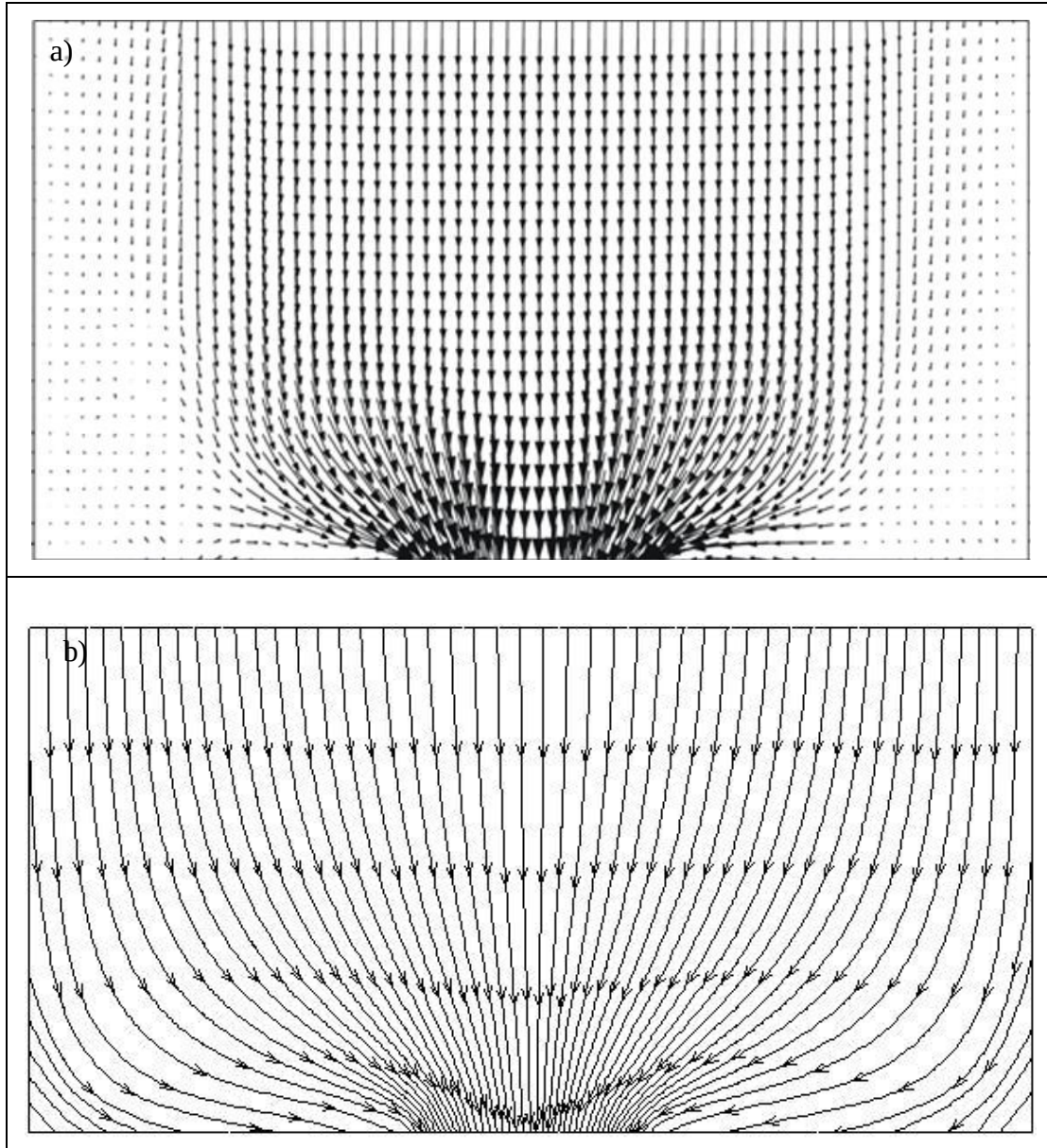


Figure 4.7. a) Streamlines of experimental study for $Re=336$
b) Streamlines of Numerical simulation for $Re=336$

When we compare the Figures 4.7a and 4.7b, we can see that the streamlines of experimental and numerical simulation are similar. Also from Figure 4.7b, we can see that the size of the vortex obtained in 4.2b is reduced as Reynolds number reaches the value of 336. Same as in Figure 4.2a, it is not easy to visualize what is happening at the corners of upstream tube.

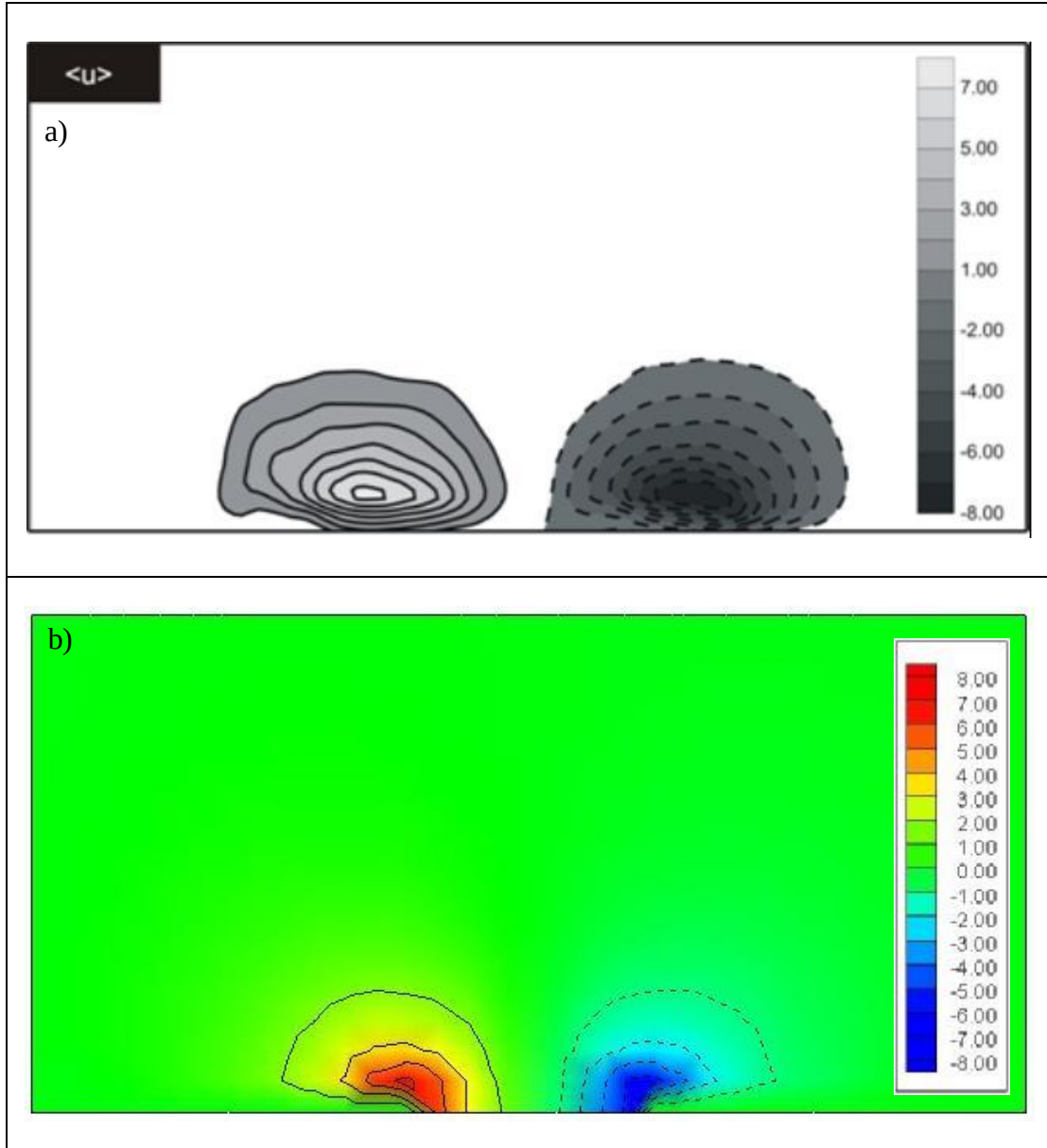


Figure 4.8. a) Average u velocity contour of PIV method for $Re=336$
 b) Average u velocity contour of numerical solution method for $Re=336$

Results of u velocity contours for Reynolds number of 336, as shown in figures above are very similar. Also when we compare the Figures 4.3 and 4.8, we can see that as the Reynolds number increases the values of velocities also increase.

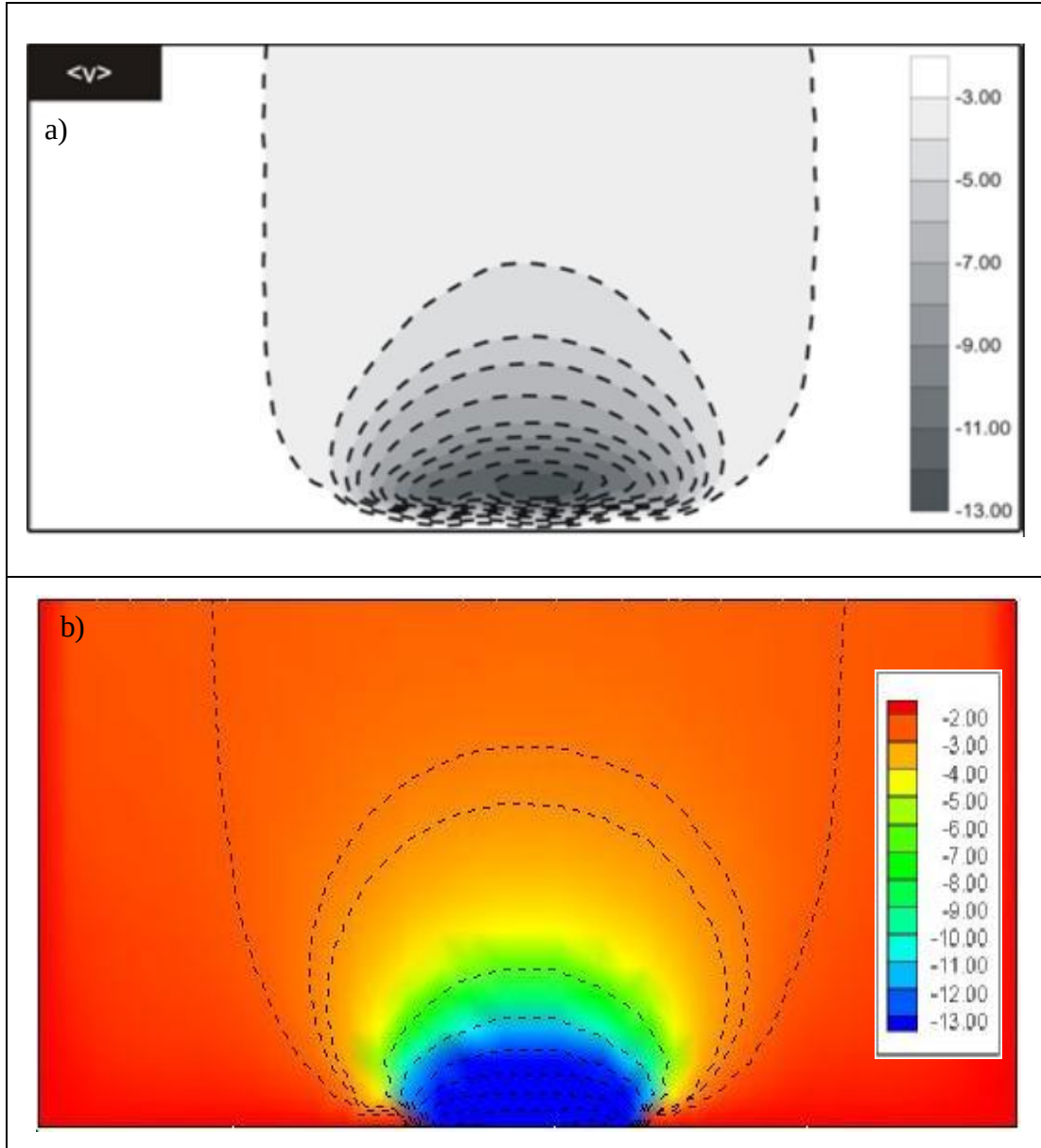


Figure 4.9. a) Average v velocity contour of PIV method for $Re=336$
 b) Average v velocity contour of numerical solution method for $Re=336$

We can see the similarities between the Figures 4.9a and 4.9b, and if we compare the Figure 4.4 and Figure 4.9 we can also see that increase in Reynolds number causes the increase in v velocity.

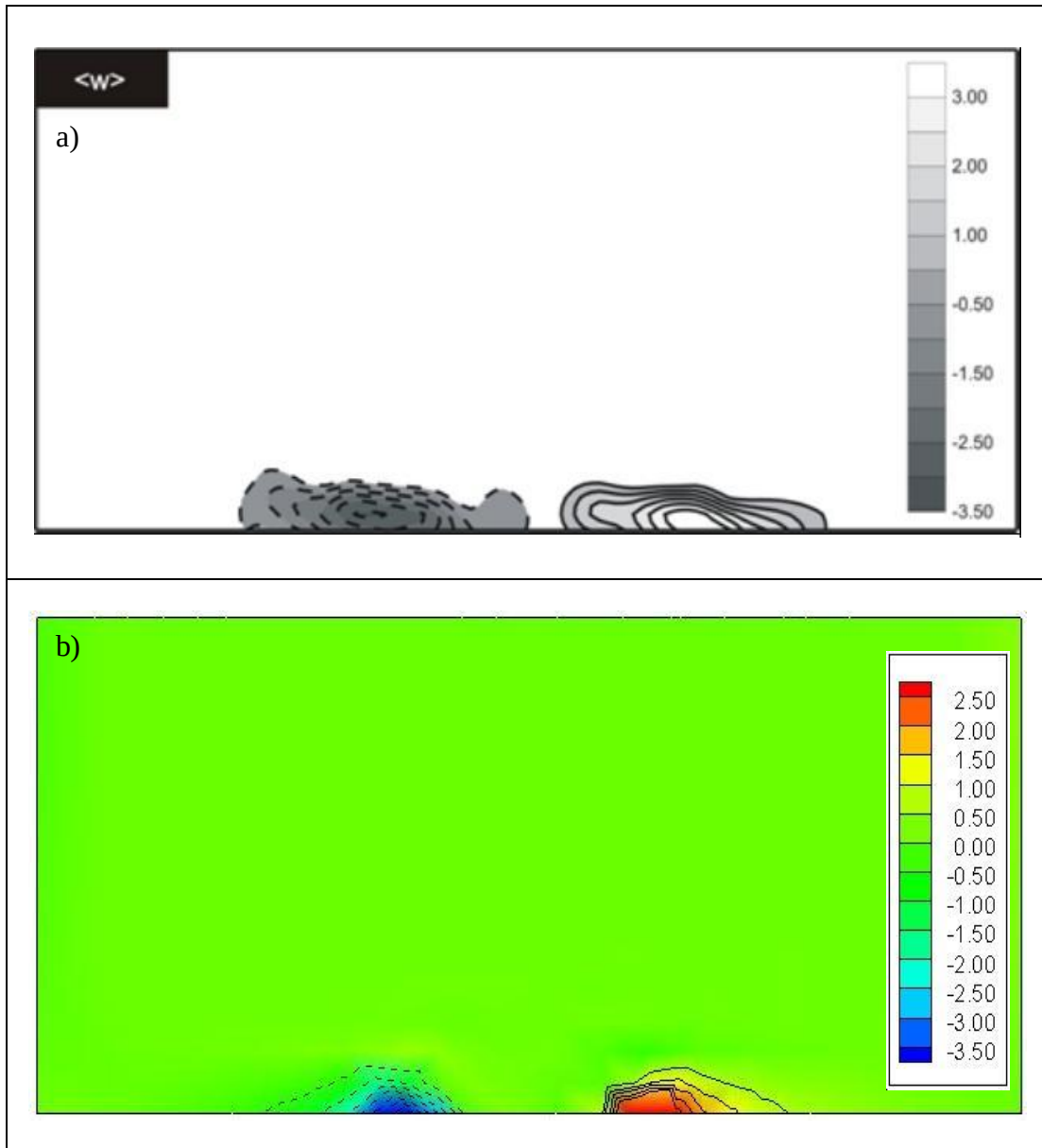


Figure 4.10. a) Average w vorticity contour of PIV method for $Re=336$

b) Average w vorticity contour of numerical solution method for $Re=336$

In Figures 4.10a and 4.10b, we see similar shapes for the contours of vorticity and same velocity scales at Reynolds number of 336.

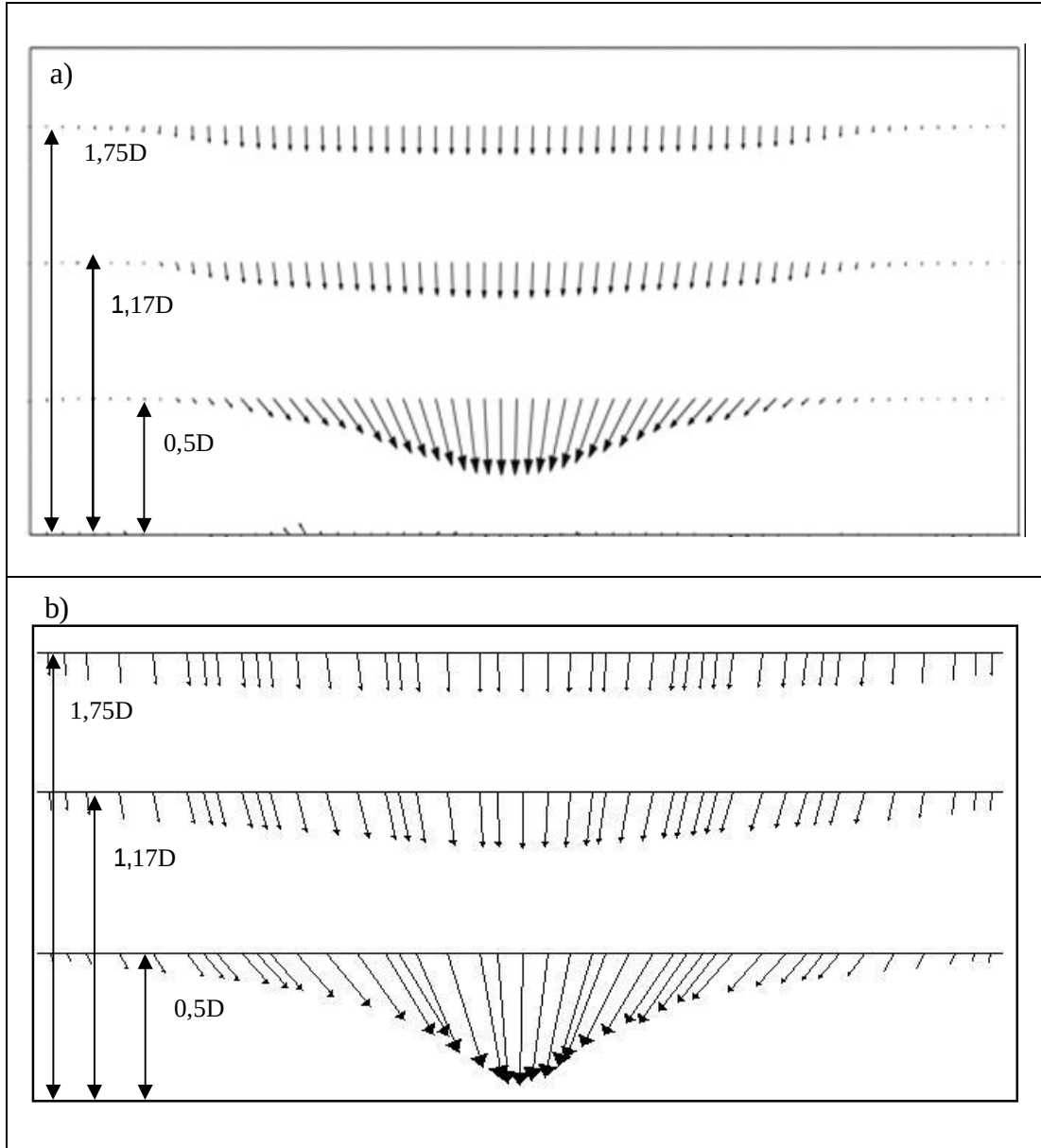


Figure 4.11. a) Average velocity vectors of PIV method at three different location for $Re=883$

b) Average velocity vectors of numerical solution at three different location for $Re=883$

As we compare the Figure 4.2, Figure 4.7 and Figure 4.11 we can observe that magnitudes of the velocity vectors continue to increase gradually together with increasing Reynolds number.

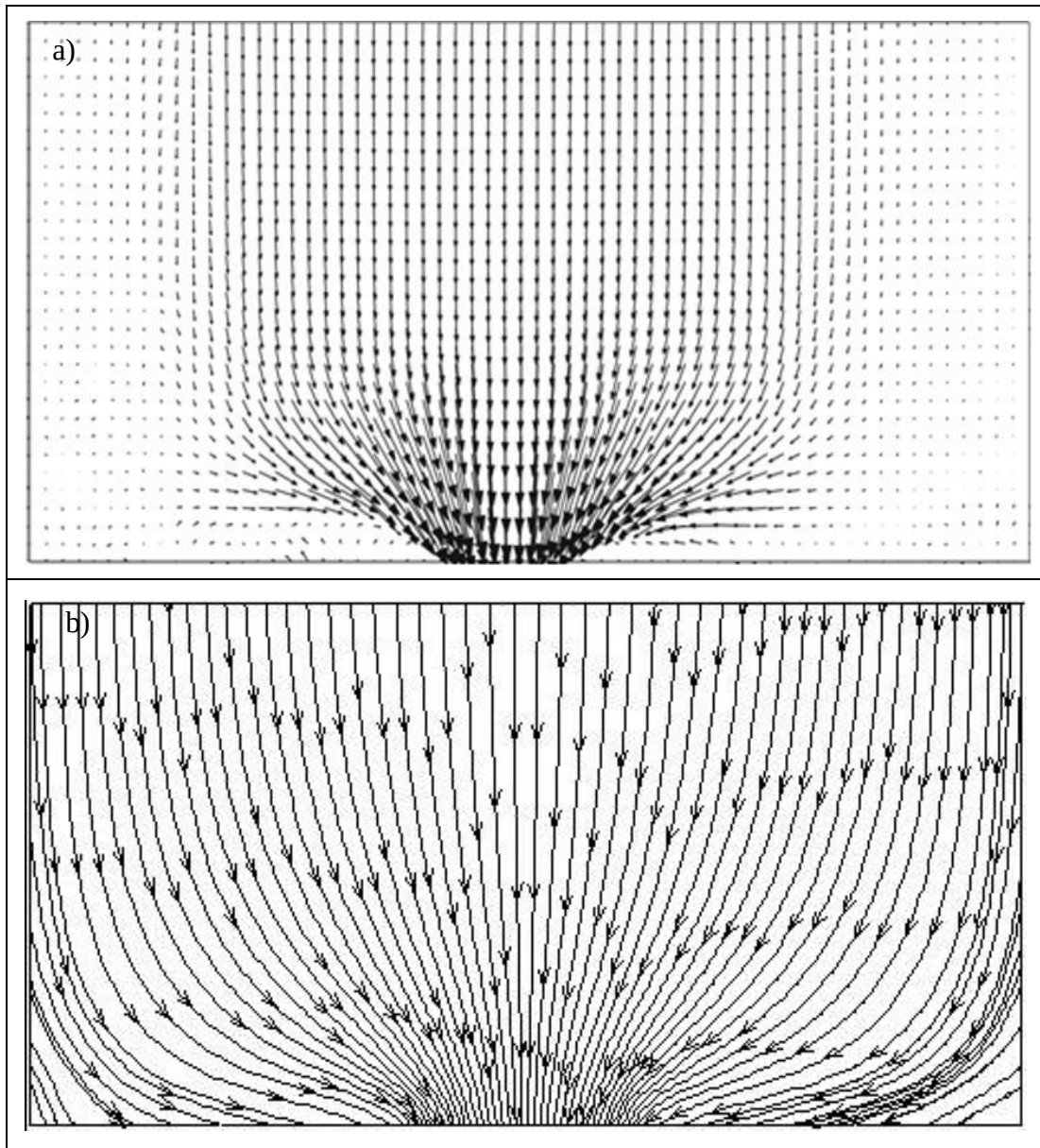


Figure 4.12. a) Streamlines of experimental study for $Re=883$
b) Streamlines of numerical simulation for $Re=883$

As we can see from Figures 4.12a and 4.12b, streamlines of experimental and numerical simulation results for Reynolds number of 883 are similar.

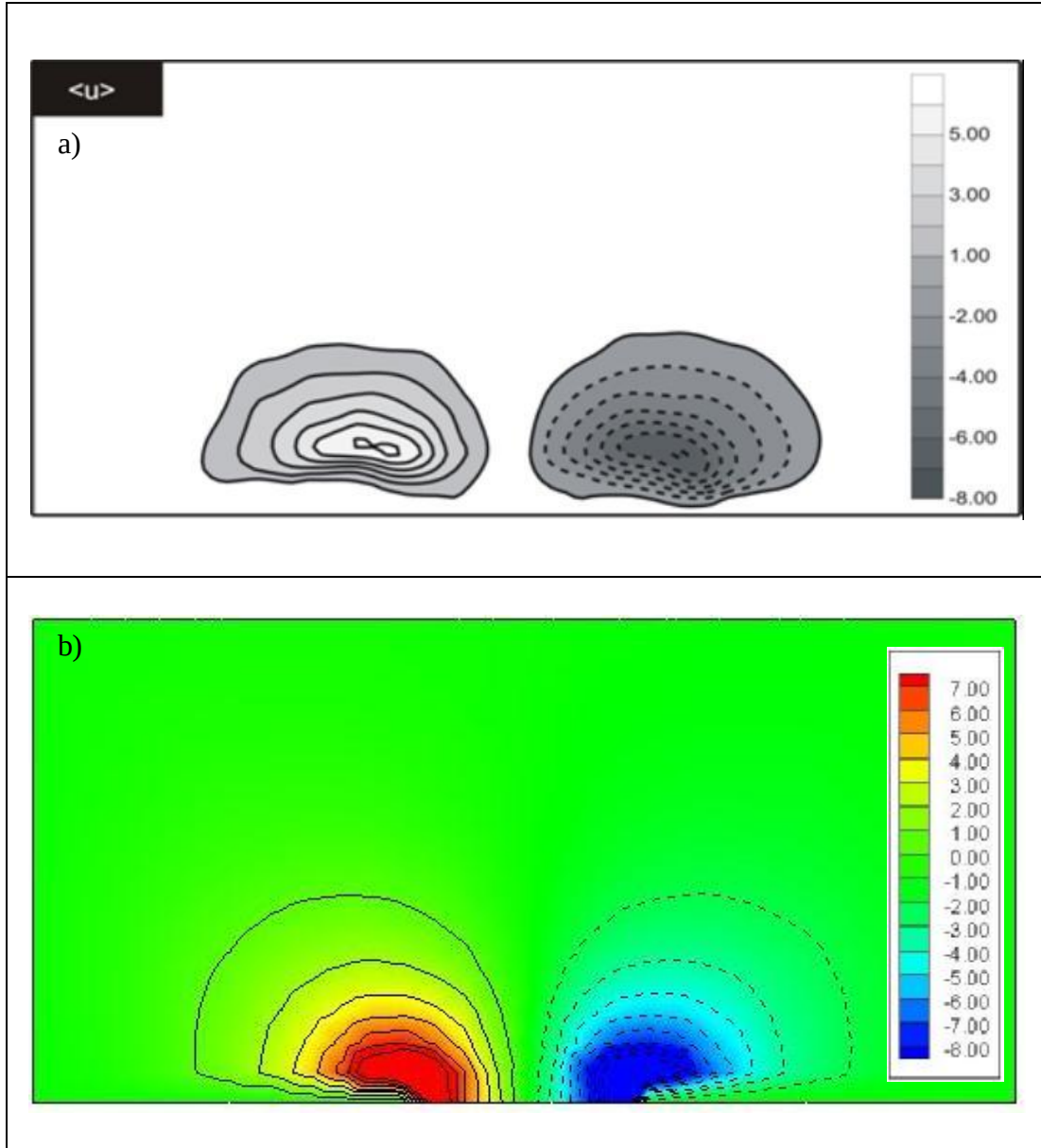


Figure 4.13. a) Average u velocity contour of PIV method for $Re=883$
 b) Average u velocity contour of numerical solution method for $Re=883$

When we compare the Figure 4.13a and Figure 4.13b, we see that both experimental and numerical methods give the same results. Also we can see from the velocity scales that values of velocity increase due to the increase of Reynolds number.

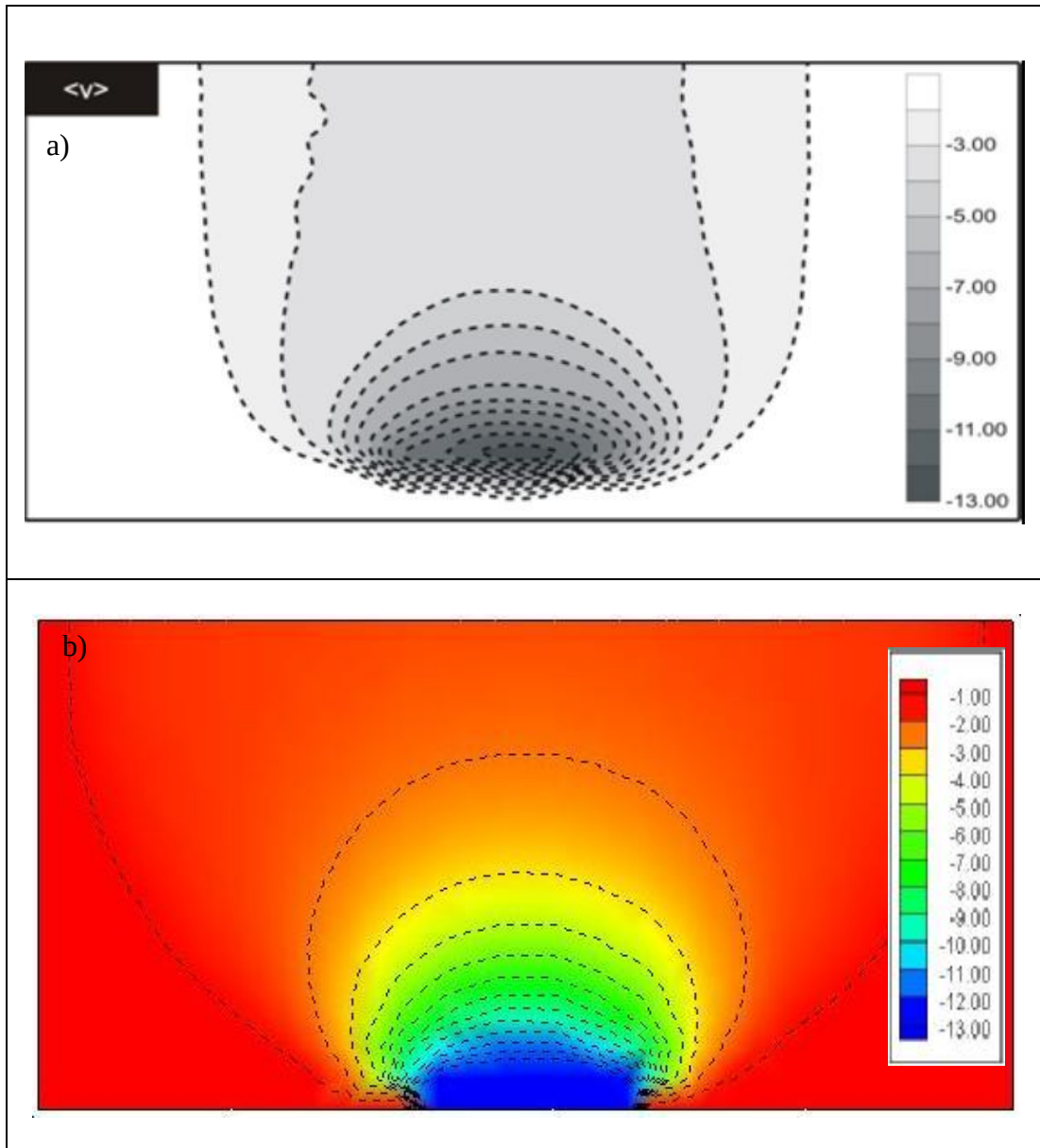


Figure 4.14. a) Average v velocity contour of PIV method for $Re=883$
b) Average v velocity contour of numerical solution method for $Re=883$

The v velocity contours for experimental and numerical simulation are very similar to each other for $Re=883$ which we can see from Figures 4.14a and 4.14b.

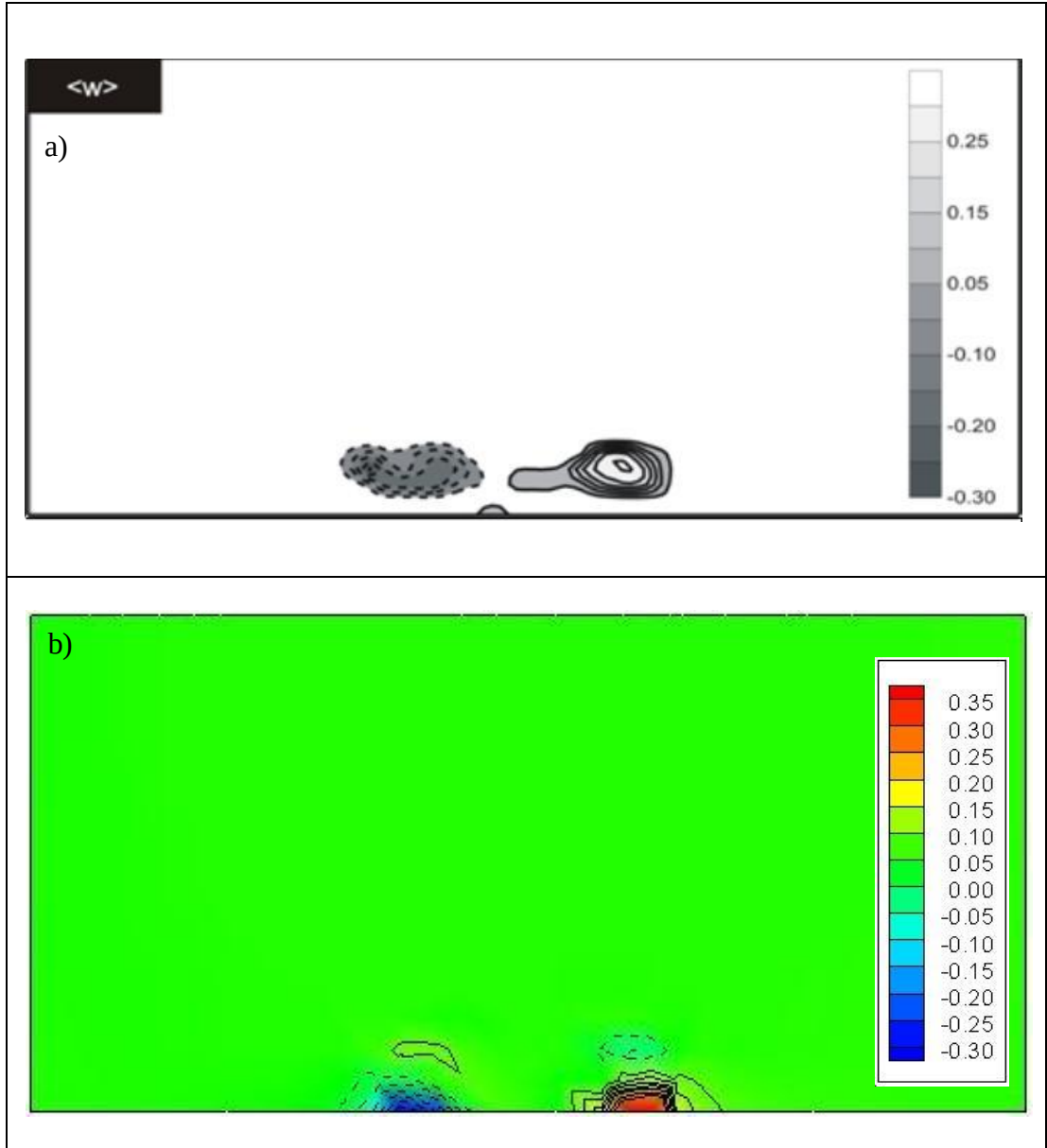


Figure 4.15. a) Average w vorticity contour of PIV method for $Re=883$

b) Average w vorticity contour of numerical solution method for $Re=883$

For the Reynolds number of 883, similar results are observed for experimental and numerical simulations, as shown in Figures 4.15a and 4.15b, respectively.

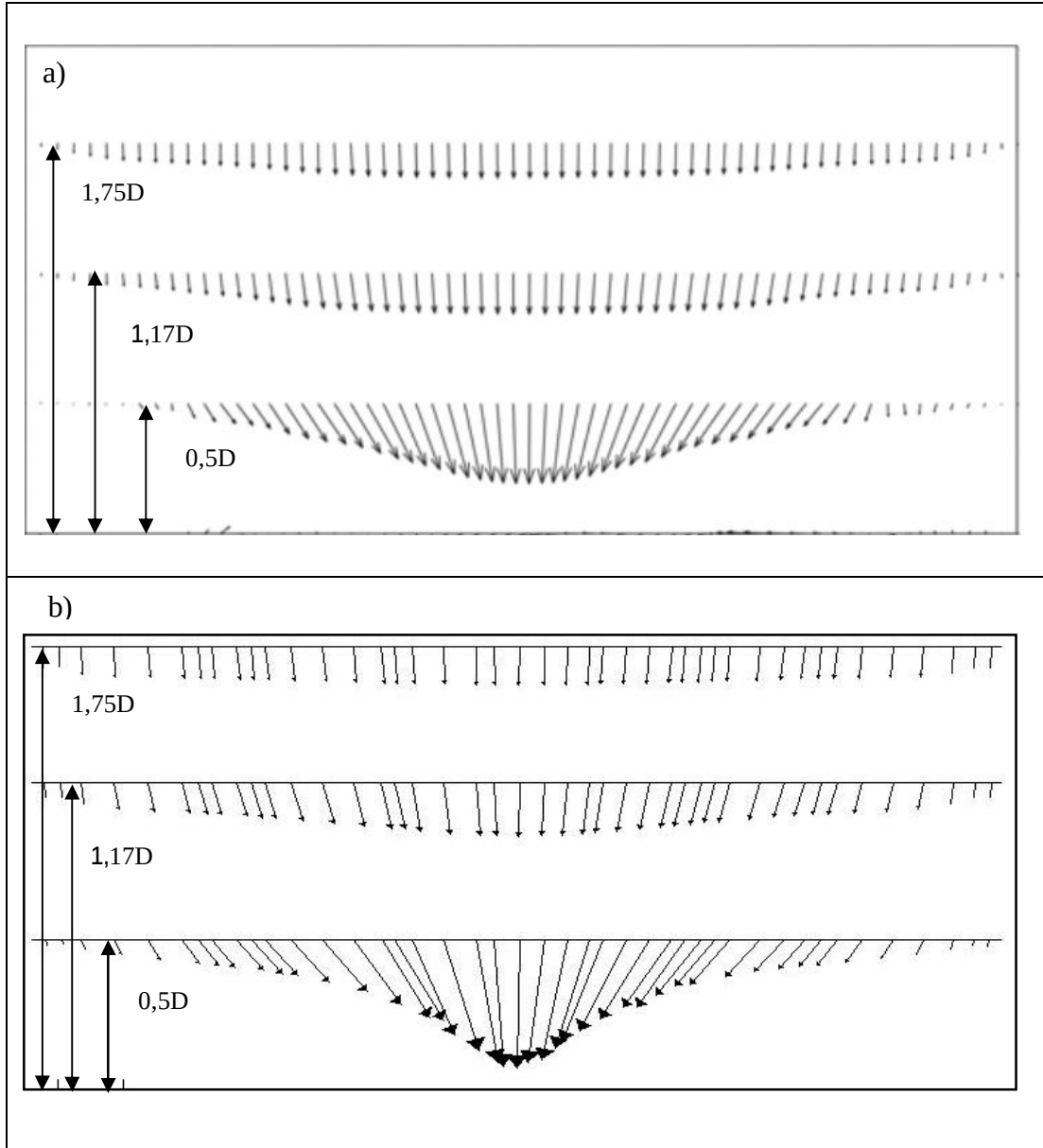


Figure 4.16. a) Average velocity vectors of PIV method at three different location for $Re=1325$

b) Average velocity vectors of numerical solution at three different location for $Re=1325$

When we compare the velocity vectors of experimental and numerical simulation for Reynolds number of 1325, we observe the same results as can be seen from Figures 4.16a and 4.16b.

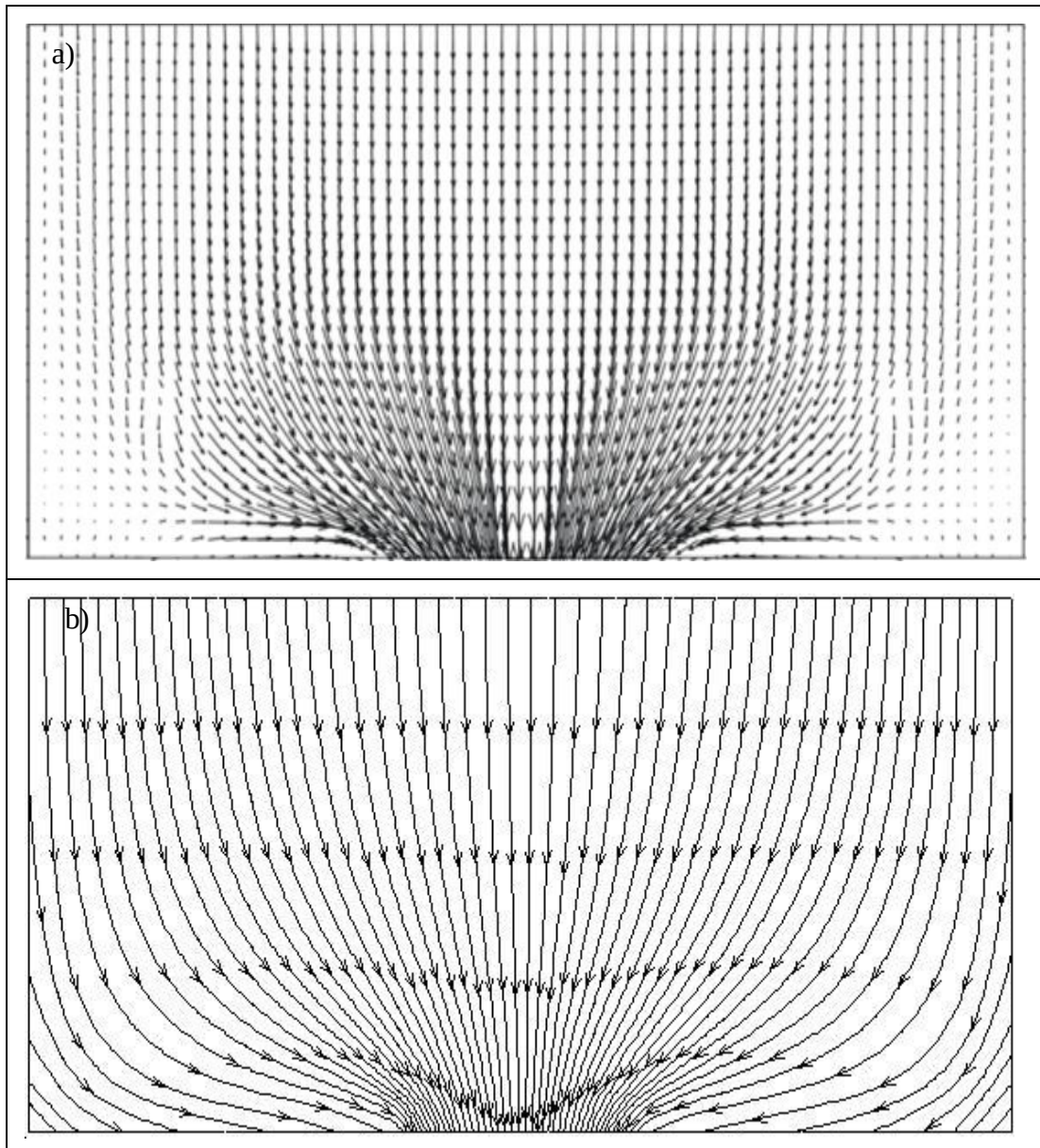


Figure 4.17. a) Streamlines of experimental study for $Re=1325$
b) Streamlines of numerical simulation for $Re=1325$

As can be seen from Figures 4.17a and 4.17b the streamlines for both experimental and numerical simulation for Reynolds number of 1325, are similar to each other.

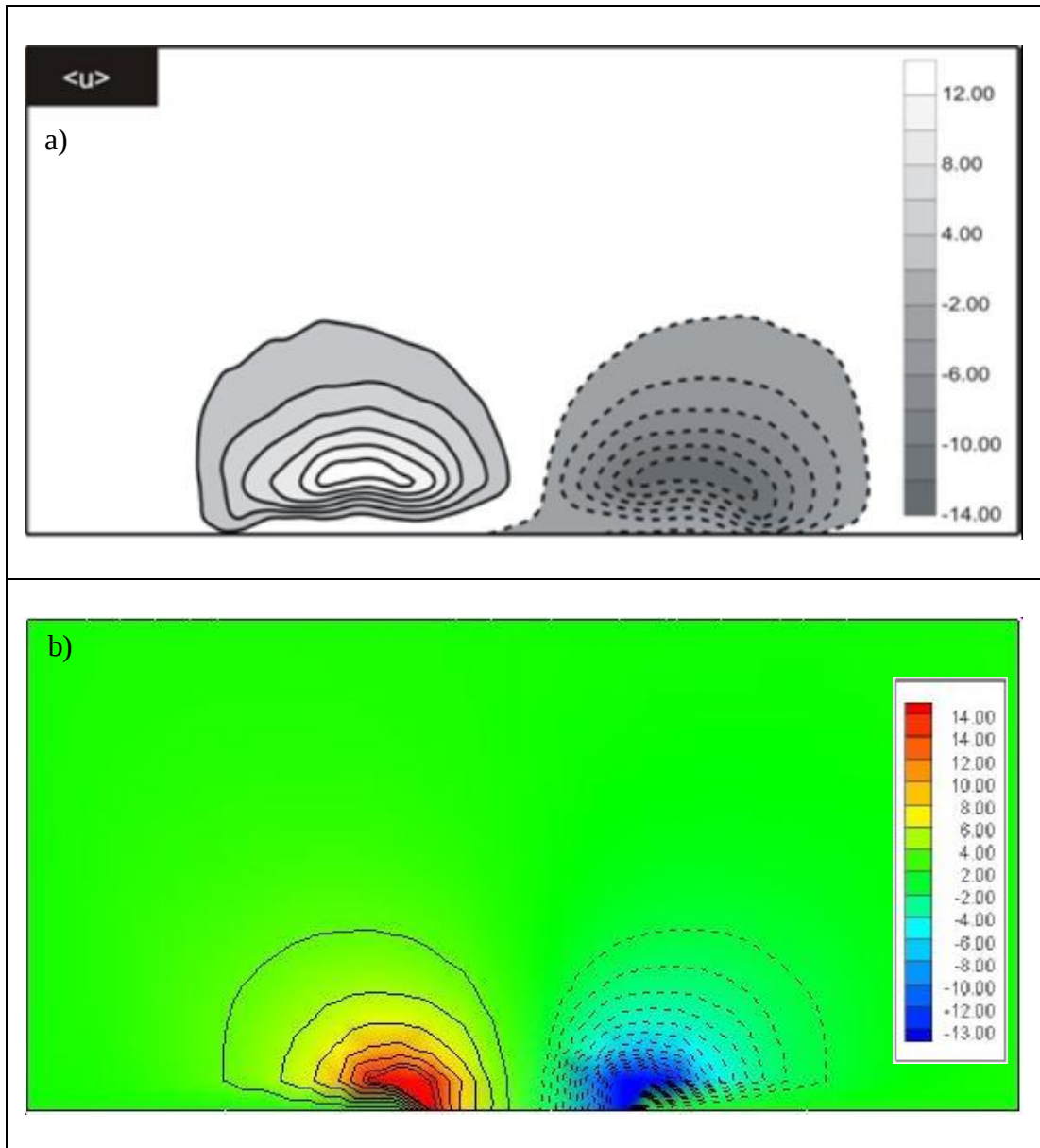


Figure 4.18. a) Average u velocity contour of PIV method for $Re=1325$.

b) Average u velocity contour of numerical solution method for $Re=1325$.

In Figures 4.18a and 4.18b we can see that the increase of velocity values according to the increase of Reynolds number.

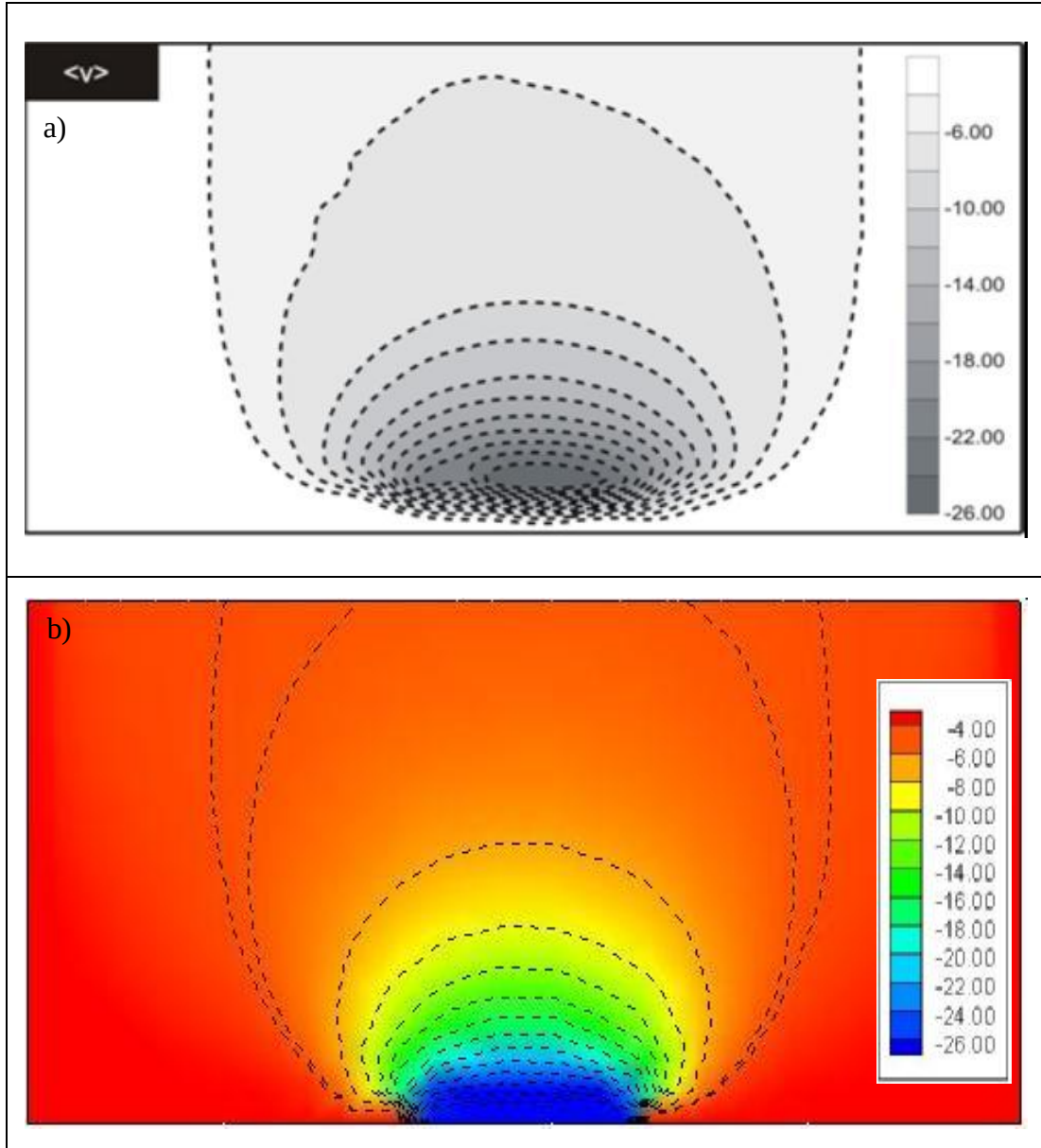


Figure 4.19. a) Average v velocity contour of PIV method for $Re=1325$
b) Average v velocity contour of numerical solution method for $Re=1325$

Same results for Reynolds number of 1325 are observed by both experimental and numerical simulation as shown in Figures 4.19a and 4.19b respectively.

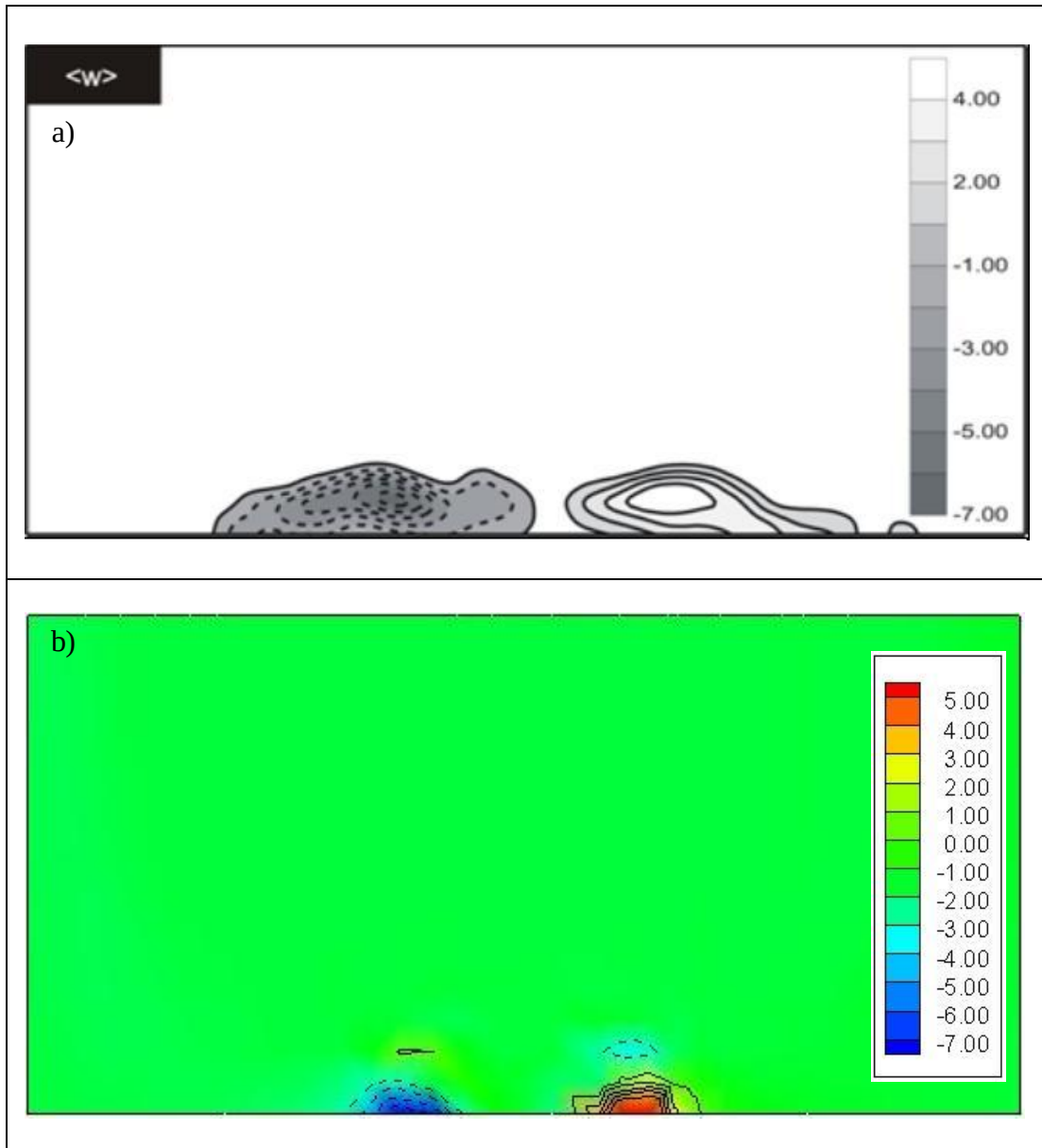


Figure 4.20. a) Average w vorticity contour of PIV method for $Re=1325$
 b) Average w vorticity contour of numerical solution method for $Re=1325$

Similar vorticity contours for Reynolds number of 1325 are shown above in Figures 4.20a and 4.20b for experimental and numerical simulation methods.

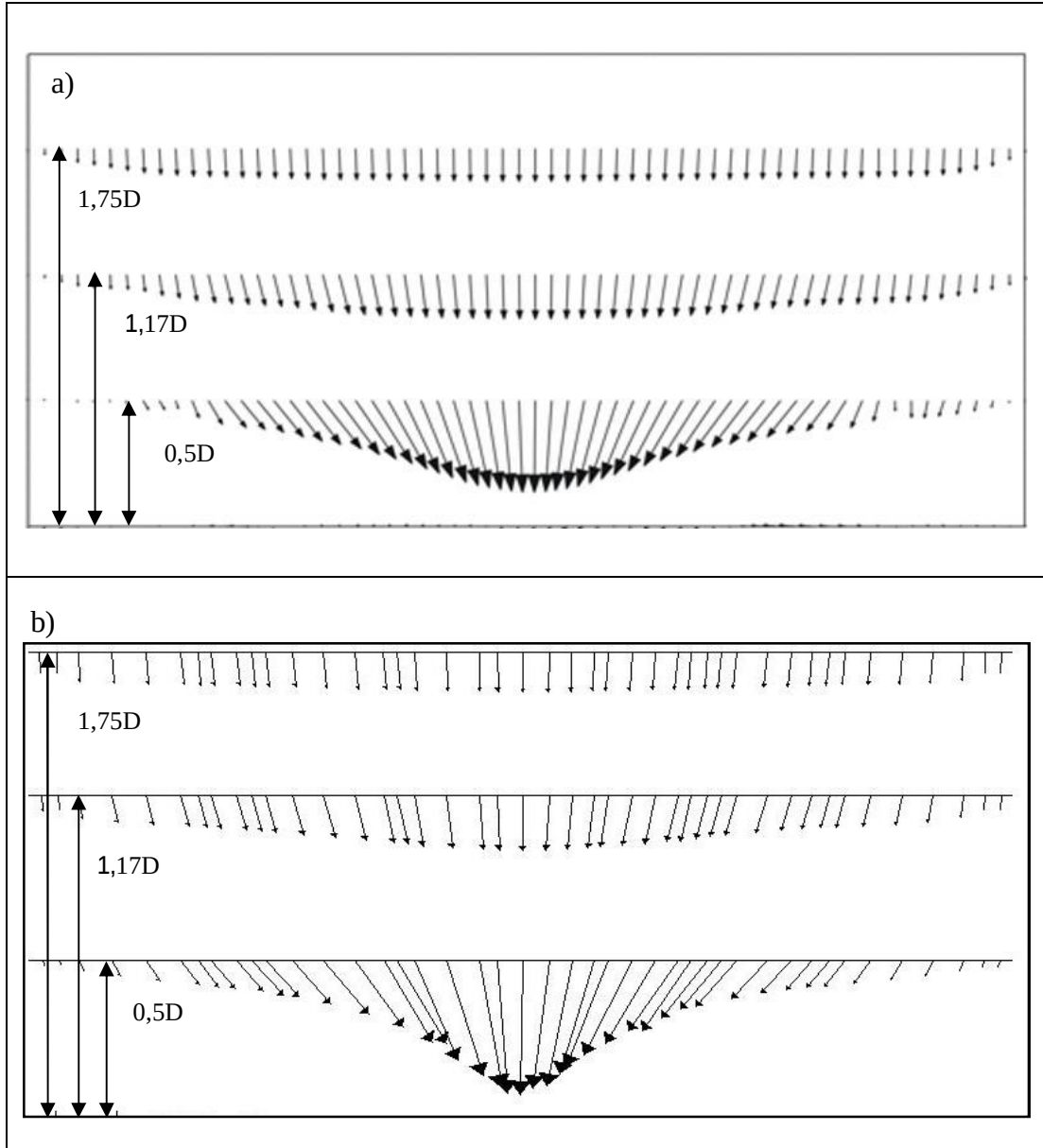


Figure 4.21. a) Average velocity vectors of PIV method at three different location for $Re=1989$

b) Average velocity vectors of numerical solution at three different location for $Re=1989$

Velocity vectors for experimental and numerical simulation are shown for Reynolds numbers from 233 up to 1989. Through these Figures we saw that general profiles and directions of the velocity vectors are very similar. Only difference is the magnitudes of the vectors which are directly related with the Reynolds number.

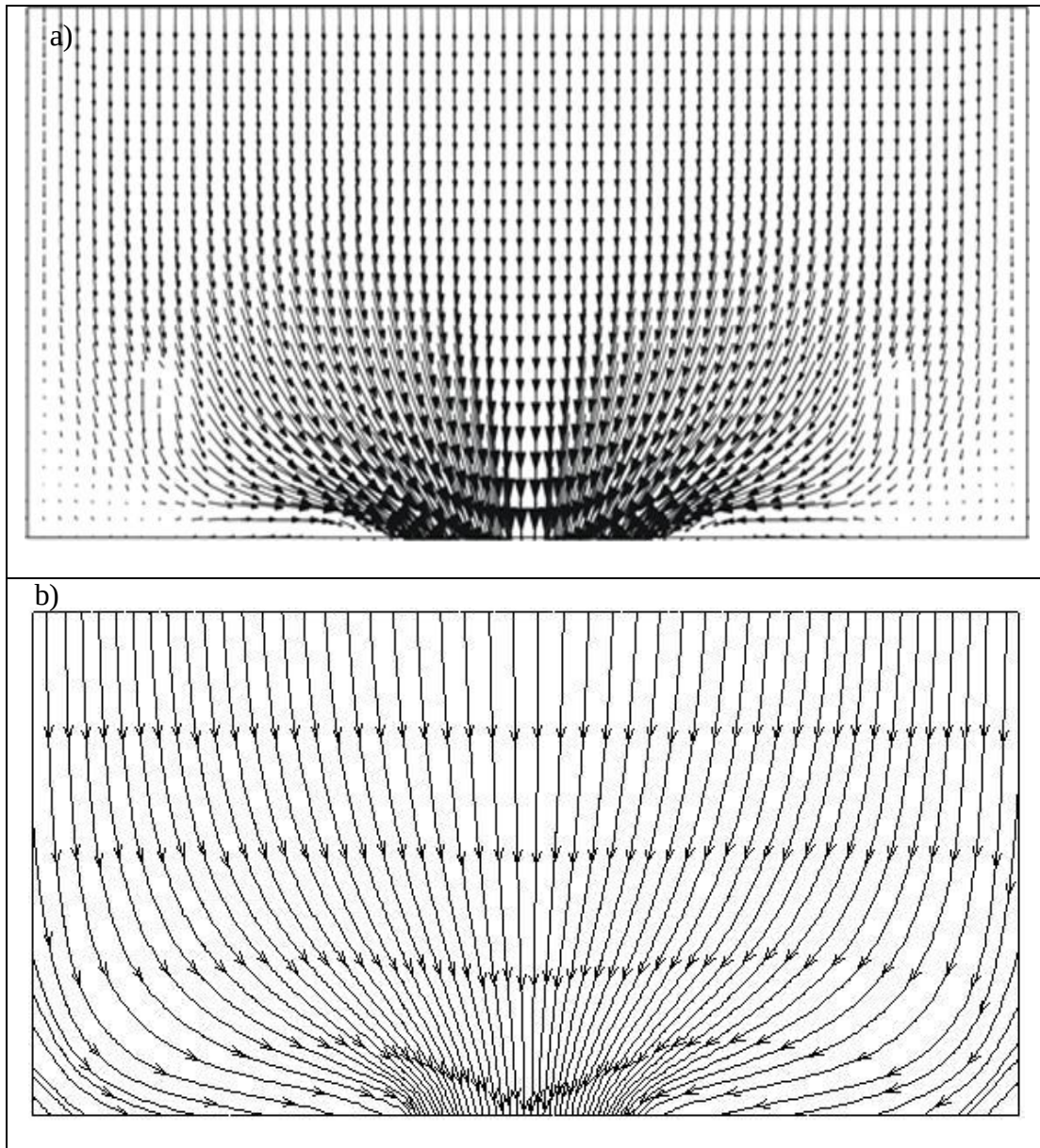


Figure 4.22. a) Streamline of experimental study for $Re=1989$
b) Streamline of numerical simulation for $Re=1989$

When we compare the streamlines of numerical simulation for Reynolds numbers from 233 through 1989, they are very similar, only difference is occurrence of vorticity just at Reynolds number of 233.

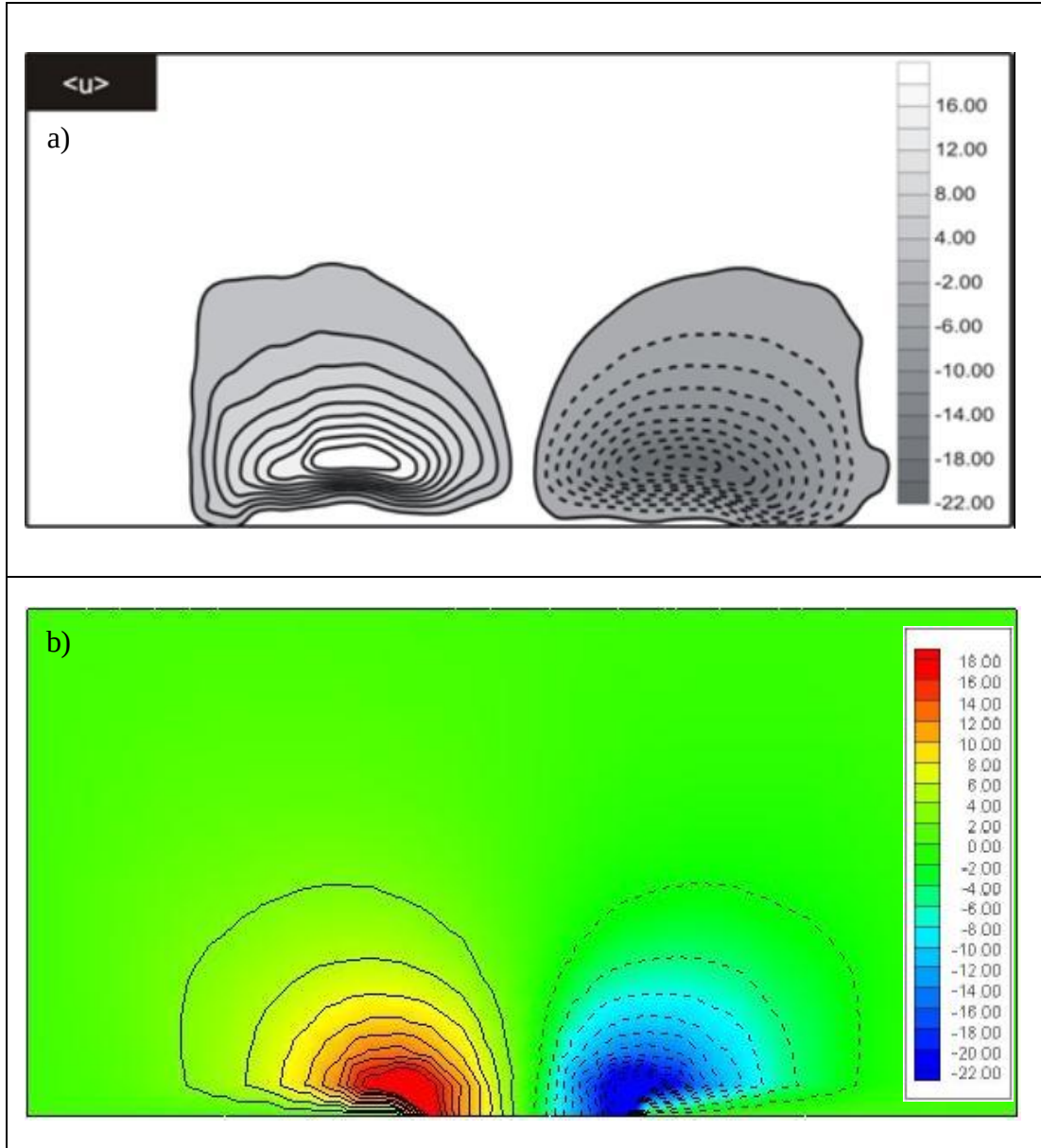


Figure 4.23. a) Average u velocity contour of PIV method for $Re=1989$

b) Average u velocity contour of numerical solution method for $Re=1989$

Average u velocity contours of experimental and numerical simulation results from Reynolds number of 233 to 1989 are very similar which show the numerical simulation is correct.

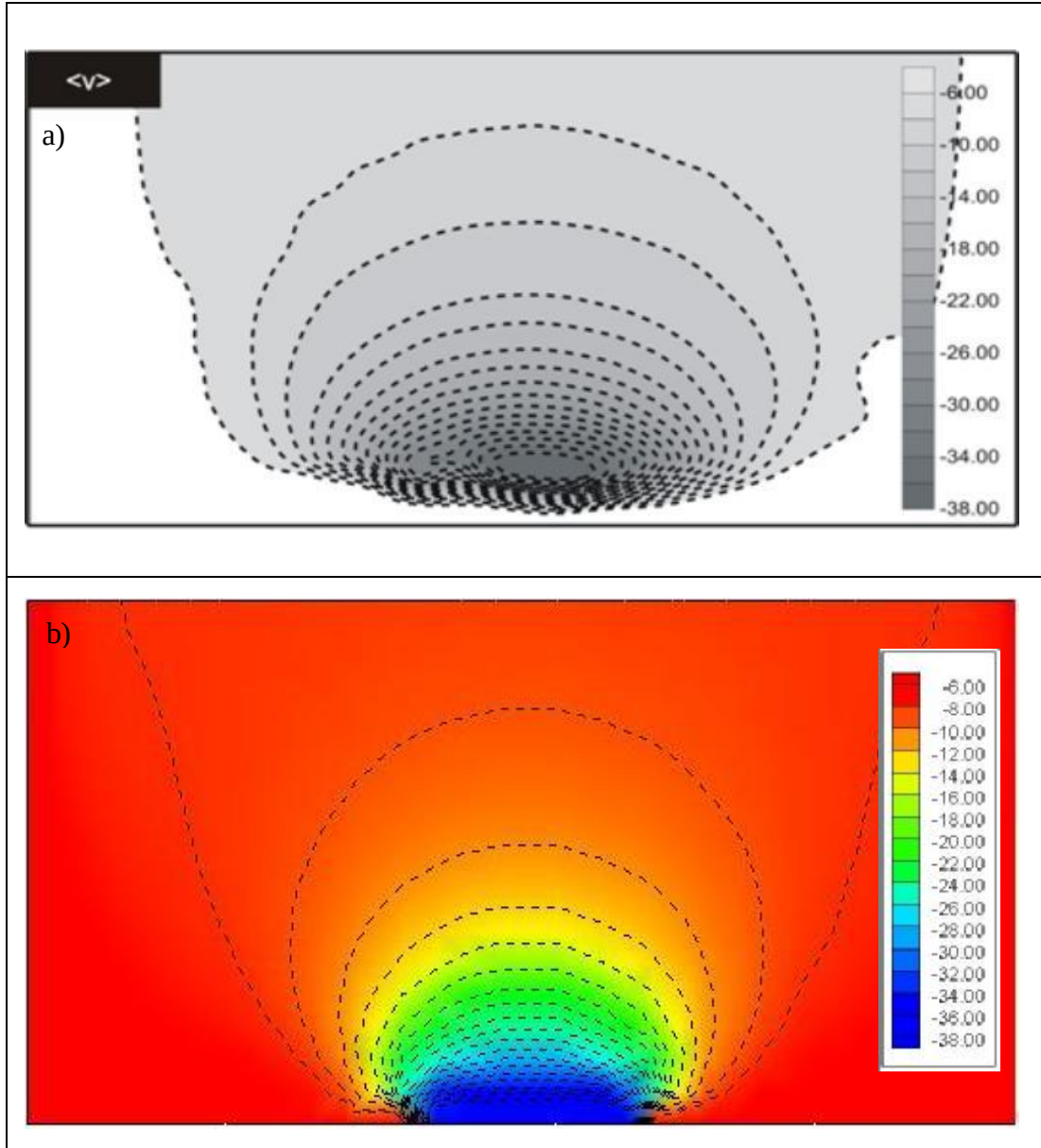


Figure 4.24. a) Average v velocity contour of PIV method for $Re=1989$
 b) Average v velocity contour of numerical solution method for $Re=1989$

From Reynolds number of 233 up to 1989 velocity contour of v showed the biggest change in shape and we saw that for all Reynolds values numerical simulation results very similarly changed the shape of v velocity contour as in experimental results.

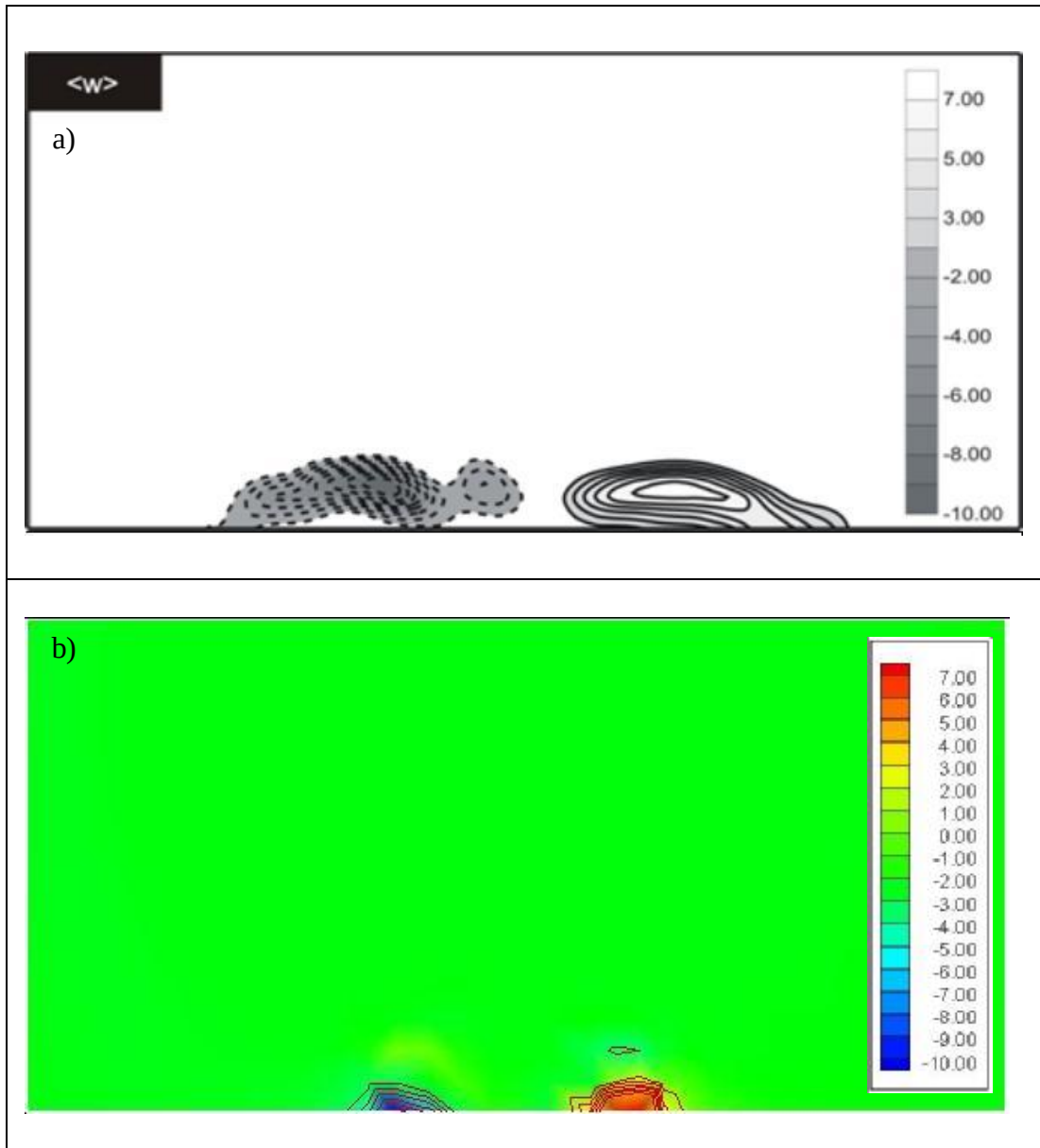


Figure 4.25. a) Average w vorticity contour of PIV method for $Re=1989$
b) Average w vorticity contour of numerical solution method for $Re=1989$

When we compare the Figures 4.25a and 4.25b we can see that same values and similar shapes of contours such as the results of the solutions at other Reynolds numbers.

4.1.2. Comparison of Axial and Radial Velocity Profiles

Figures 4.26 through 4.35 represents the V_x and V_y profiles at different y/D ratios and Reynolds numbers. The V_x charts are plotted for y/D ratios of 0.36, 0.43, 0.51 and 0.73 as show on the right corners. The V_y charts are plotted for y/D ratios of 0.29, 0.43, 0.58 and 0.95 also shown in right corners.

Figure numbers with an extension letter “a” (like in Figs. 4.26a, 4.35a) show the velocity profiles of the experimental results of Fakilar, M.S., (2004) and the figure numbers with an extension letter “b” (like in Figs. 4.26b, 4.35b) show the velocity profiles of the numerical simulation. When the experimental velocity profiles are compared with the numerical simulation velocity profiles, both show the same characteristics.

For V_x profiles, in both results the value of radial velocity is negative on right, positive on left and zero in the centre of flow. When y/D ratio increases the magnitude of the profile decreases as can be seen from the Figures. As we increase the Reynolds number from 233 to 1989 the magnitude of the profiles also increases.

For V_y profiles, in both of them, value of V_y increases when one goes away from tube wall and reaches the maximum value at the centre of flow and then decreases again as it gets closer to the tube wall. As we can see from both Figures the value of V_y increases from 20 mm to 50 mm. Also it can be seen from the Figures when flow is close to the contraction plane which means reducing the y/D ratio, V_y values increases that means flow accelerates. When we increase the Reynolds number the V_y values also increase.

Although the values of v velocity contours are negative because of choosing the upside direction as positive for y axis in numerical simulation software, all V_y velocities that we observed in these profiles are positive, because absolute values are taken.

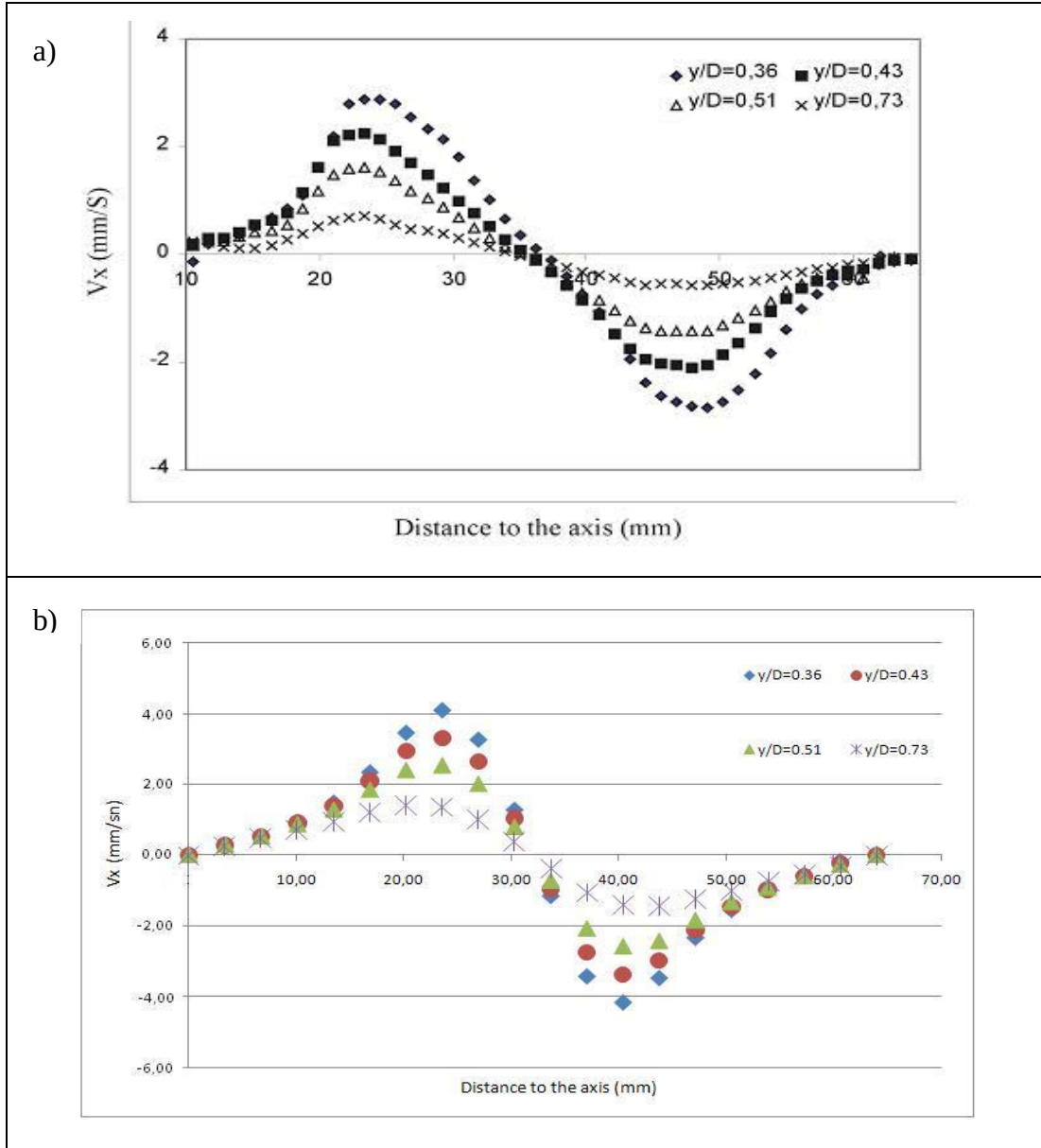


Figure 4.26. a) Experimental result for radial velocity profiles V_x at $Re=233$
 b) Numerical simulation result for radial velocity profiles V_x at $Re=233$

Figures 4.26a and 4.26b show the V_x velocity profiles of experimental and numerical simulation respectively for Reynolds number of 233. As we can see from figures V_x profiles are very similar in both study.

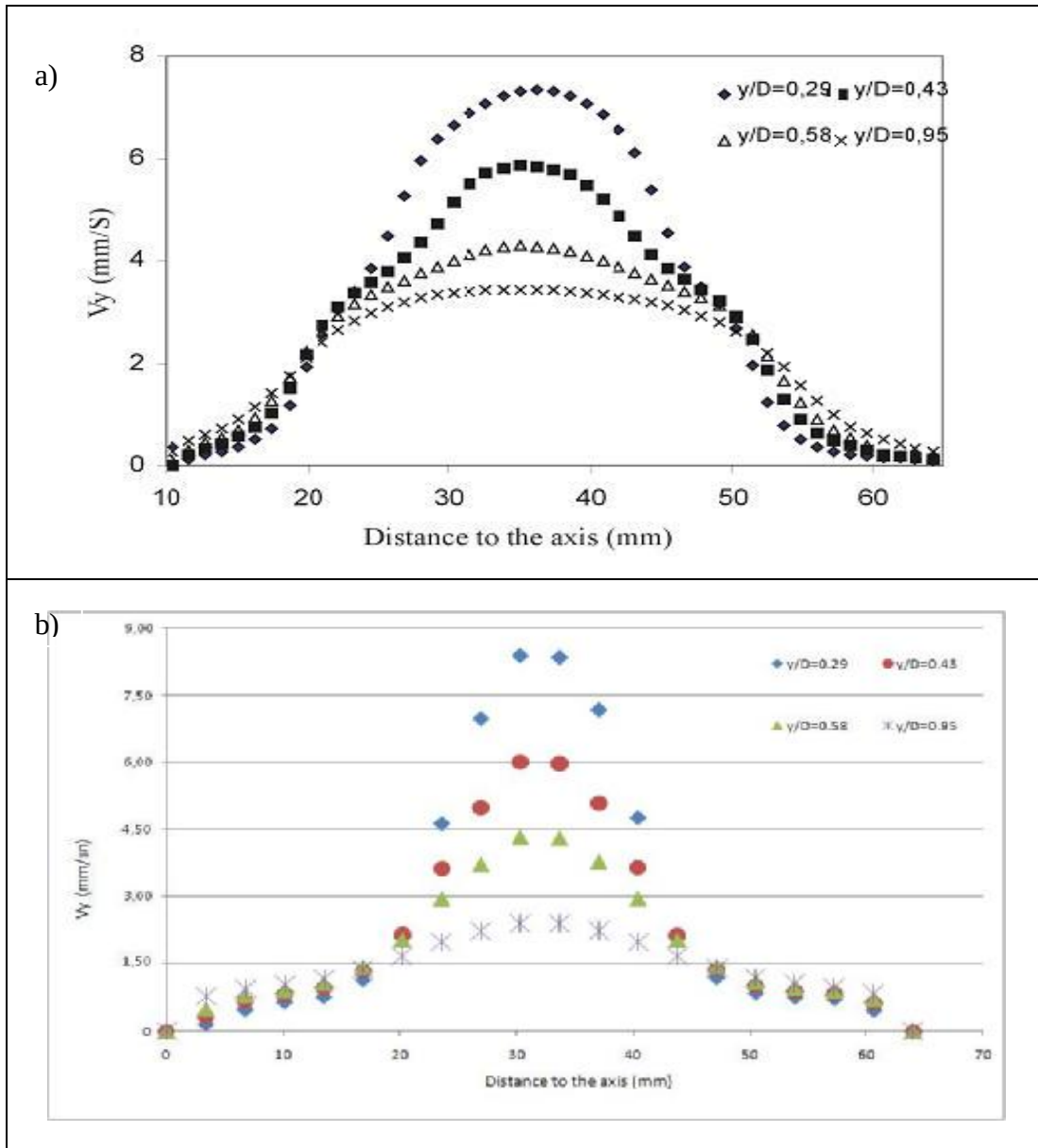


Figure 4.27. a) Experimental result for axial velocity profiles V_y at $Re=233$
 b) Numerical simulation result for axial velocity profiles V_y at $Re=233$

Experimental and numerical simulation results of V_y velocity for Reynolds number of 233 are shown in Figures above. As we can see profiles are similar. Difference is that after the distance of 20 mm V_y velocity profile of numerical simulation increases more sharply when compared with the experimental result.

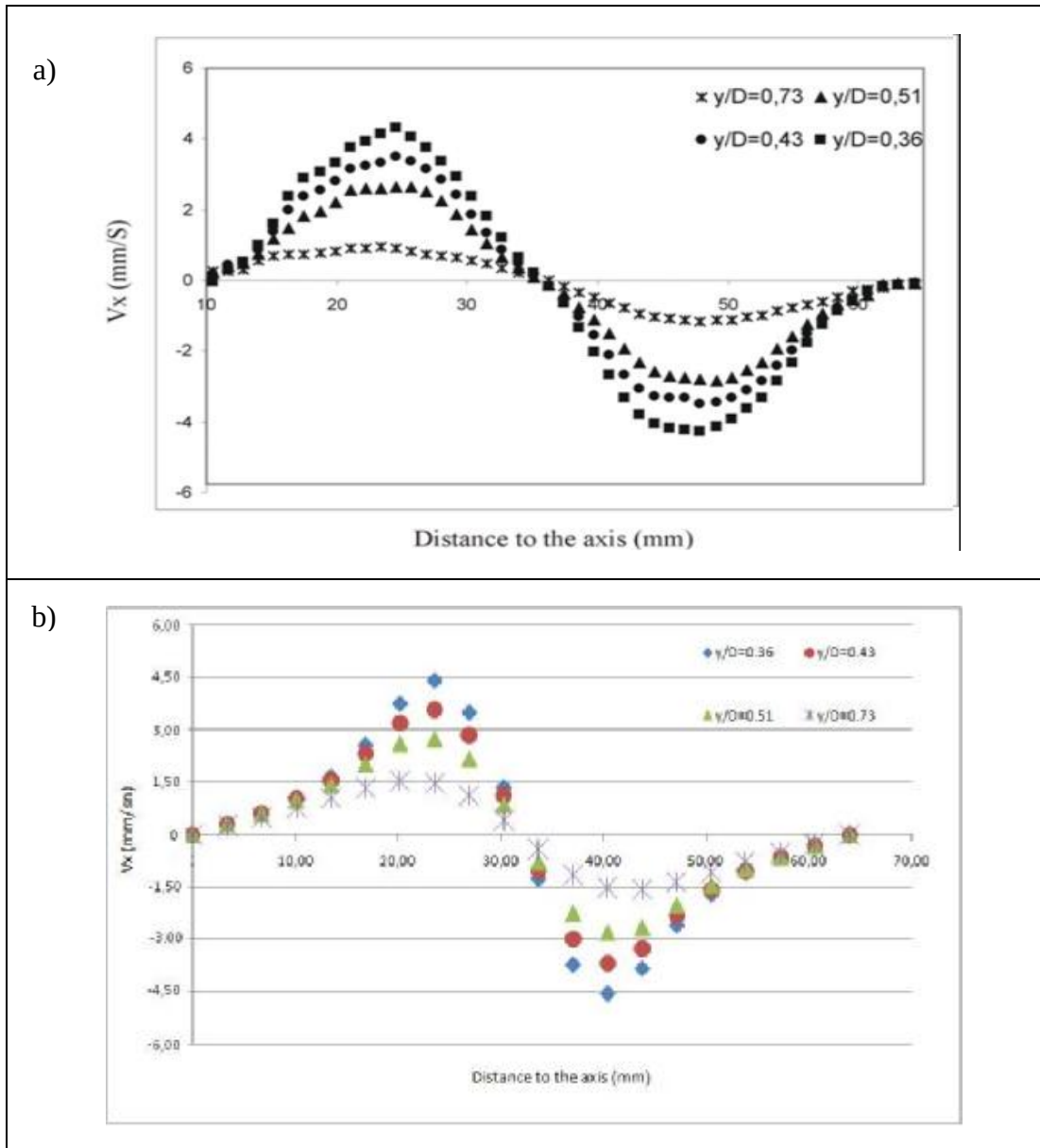


Figure 4.28. a) Experimental result for radial velocity profiles V_x at $Re=336$
 b) Numerical simulation result for radial velocity profiles V_x at $Re=336$

For Reynolds number of 336, V_x profiles of both study is the same. Also when we compare the Figures 4.26 and 4.28, we can see that minimum and maximum values are higher in Figure 4.28 as a result of increasing Reynolds number.

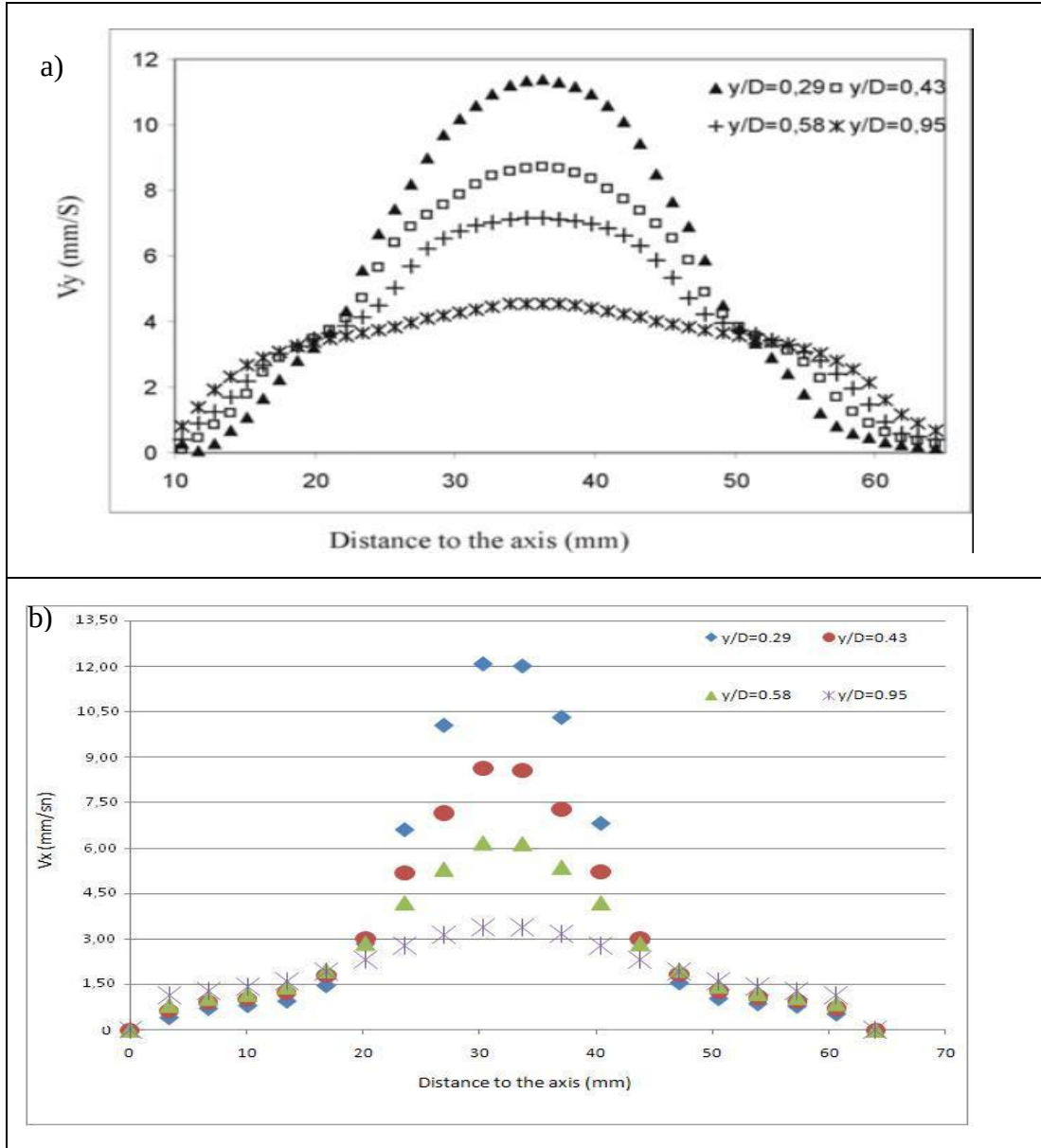


Figure 4.29. a) Experimental result for axial velocity profiles V_y at $Re = 336$

b) Numerical simulation result for axial velocity profiles V_y at $Re = 336$

In Figures 4.29a and 4.29b we can see the similar V_y profiles for both study. Same as in V_x profiles, maximum values of V_y profiles in Figure 4.29 is higher than Figure 4.27 due to increasing Reynolds number.

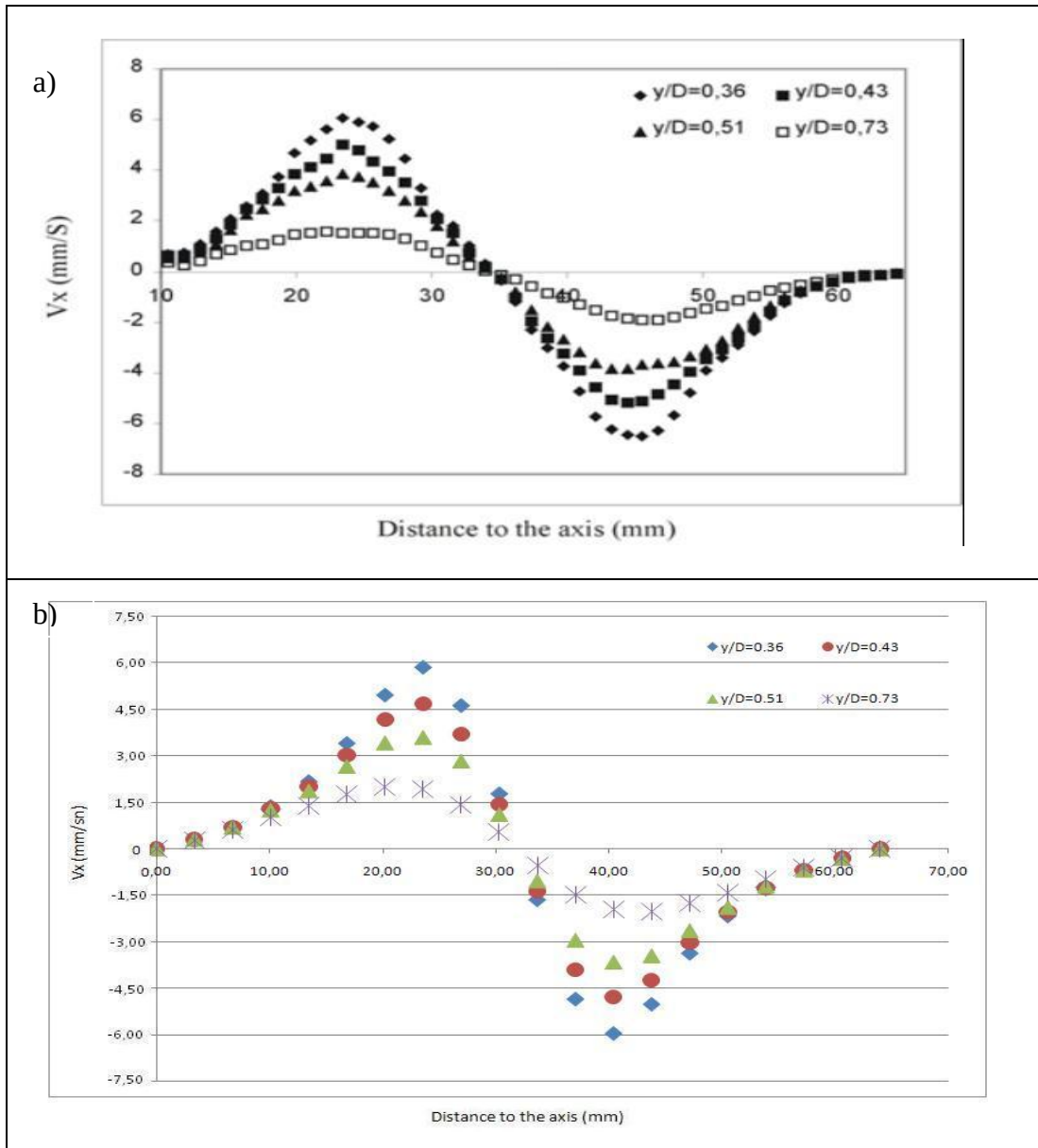


Figure 4.30. a) Experimental result for radial velocity profiles V_x at $Re = 883$
 b) Numerical simulation result for radial velocity profiles V_x at $Re = 883$

Figures 4.30a and 4.30b show the same profiles of V_x velocity. We can also see the effect of increasing Reynolds number.

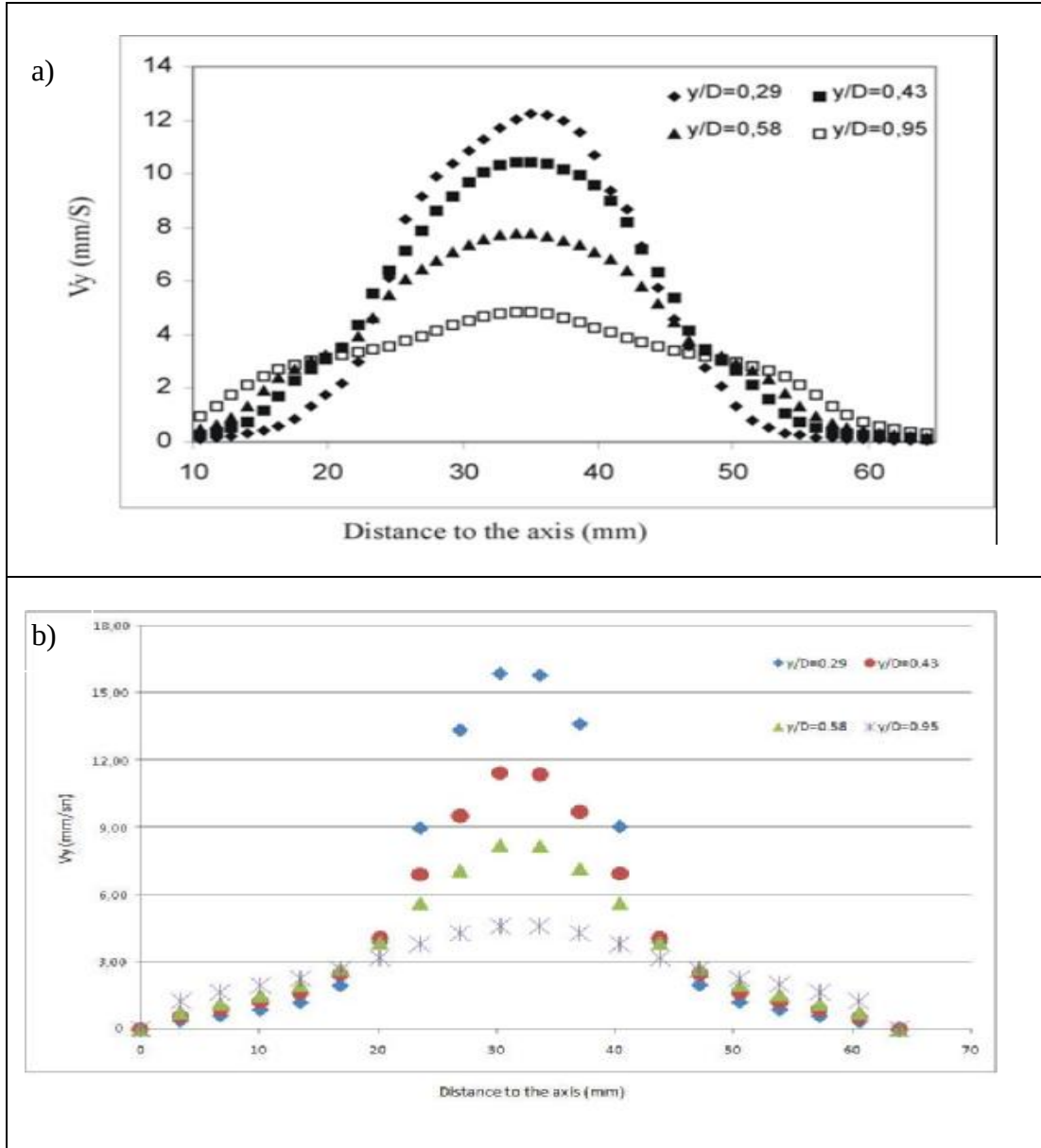


Figure 4.31. a) Experimental result for axial velocity profiles V_y at $Re=883$

b) Numerical simulation result for axial velocity profiles V_y at $Re=883$

V_y velocity profiles along the diameter show the same characteristics for both study at Reynolds number of 883, as can be seen from Figures above.

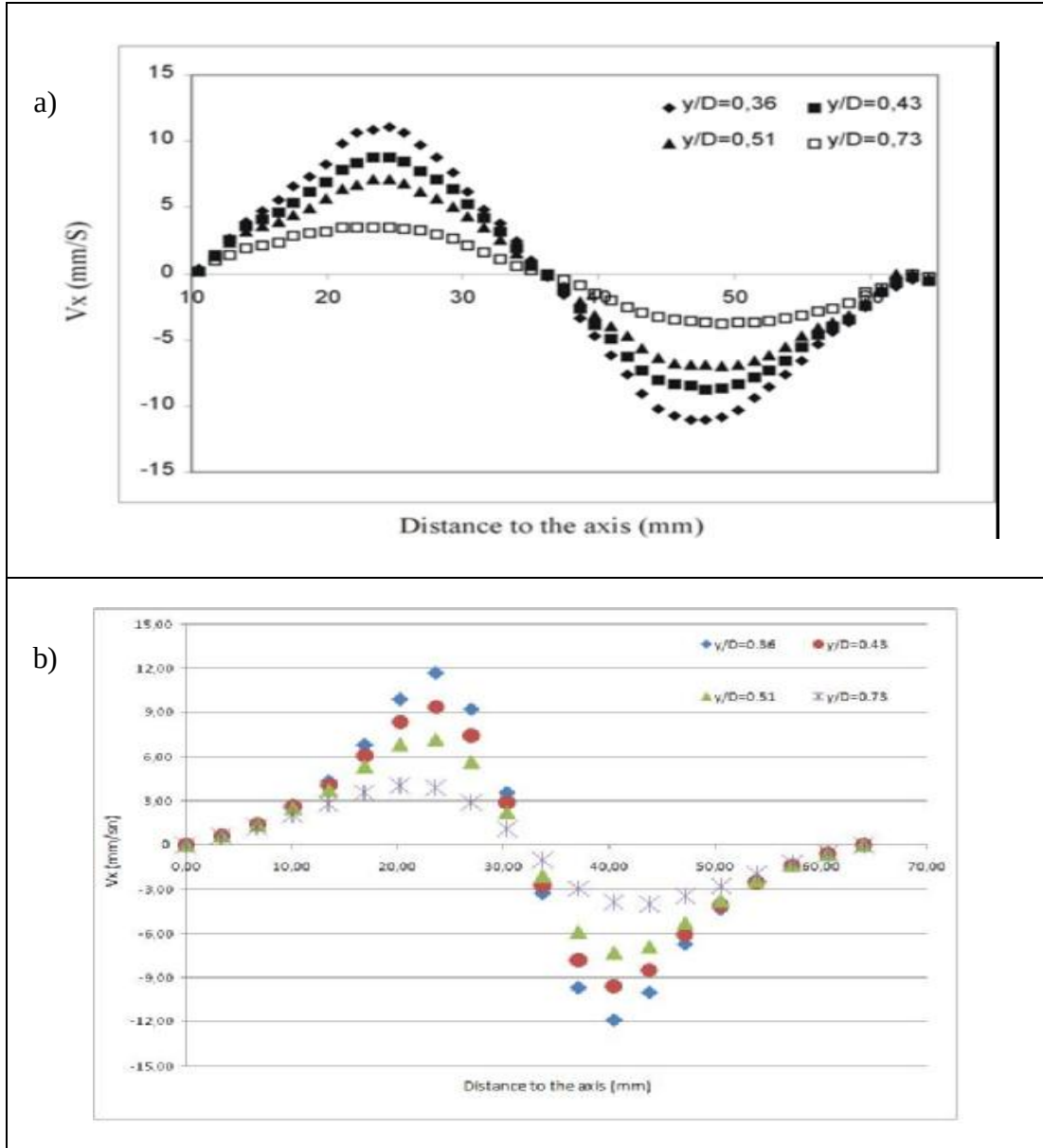


Figure 4.32. a) Experimental result for radial velocity profiles V_x at $Re= 1325$
 b) Numerical simulation result for radial velocity profiles V_x at $Re= 1325$

Figures 4.32a and 4.32b show profiles observed for previous Reynolds numbers, only difference is the maximum and minimum values.

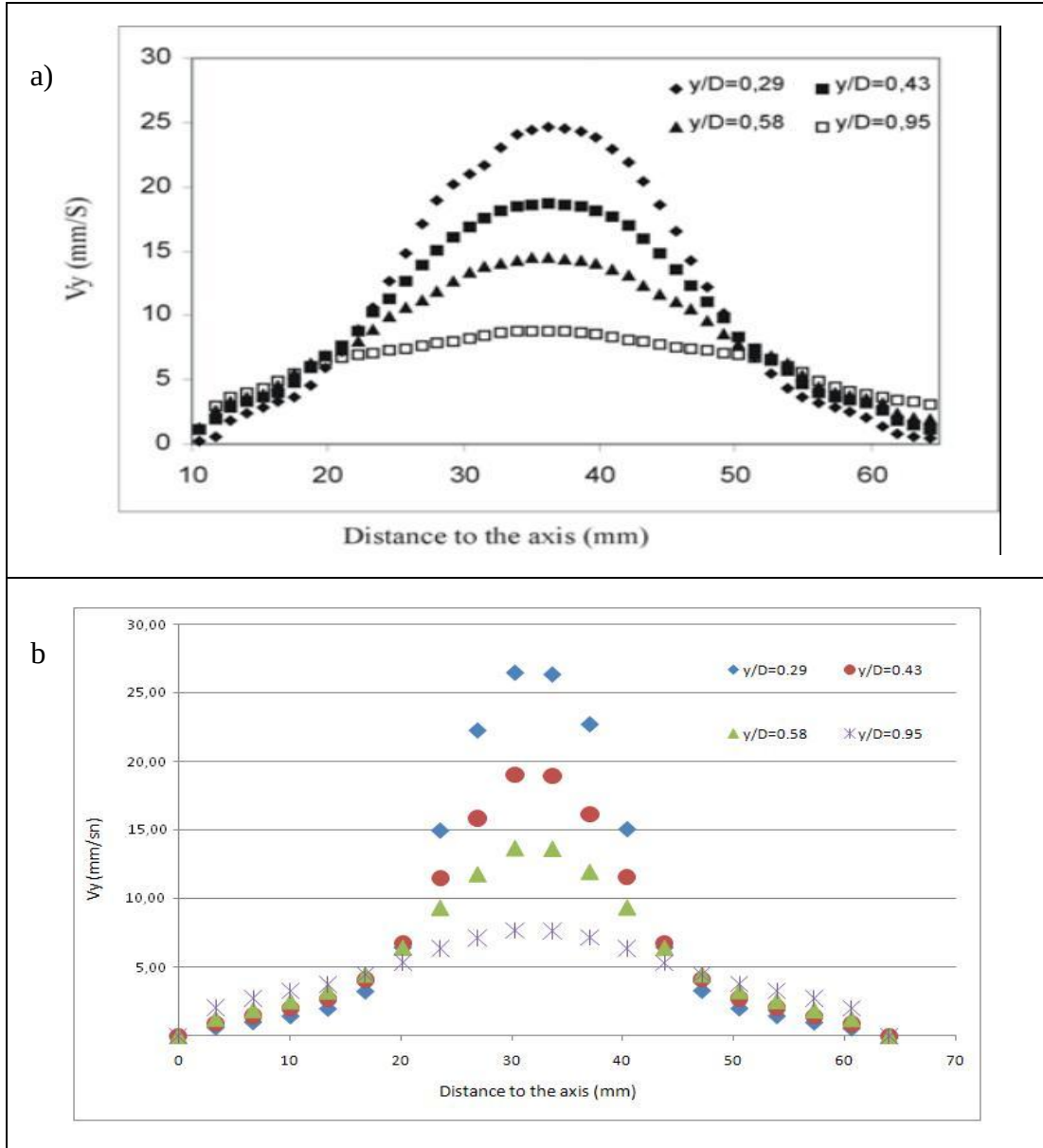


Figure 4.33. a) Experimental result for axial velocity profiles V_y at $Re=1325$

b) Numerical simulation result for axial velocity profiles V_y at $Re=1325$

Experimental and numerical simulation results for Reynolds number of 1325 show the same V_y profile as can be seen from Figures above.

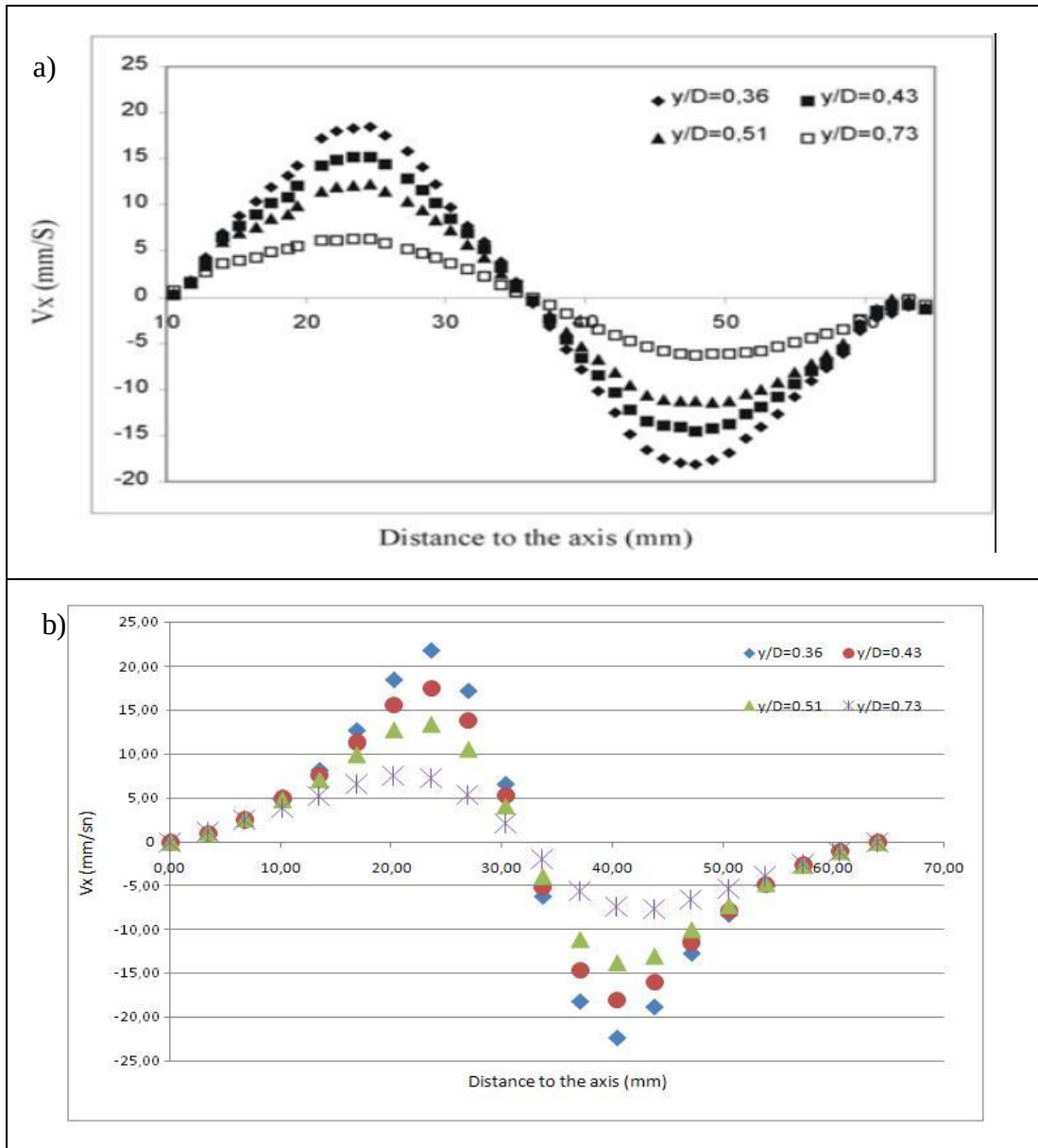


Figure 4.34. a) Experimental result for radial velocity profiles V_x at $Re = 1989$
 b) Numerical simulation result for radial velocity profiles V_x at $Re = 1989$

From Reynolds number of 233 to 1989 we observed the same profiles for V_x velocity, that is what we already expected according to velocity contours discussed before. As we can see the only difference is the maximum and minimum values which is directly related with the Reynolds number.

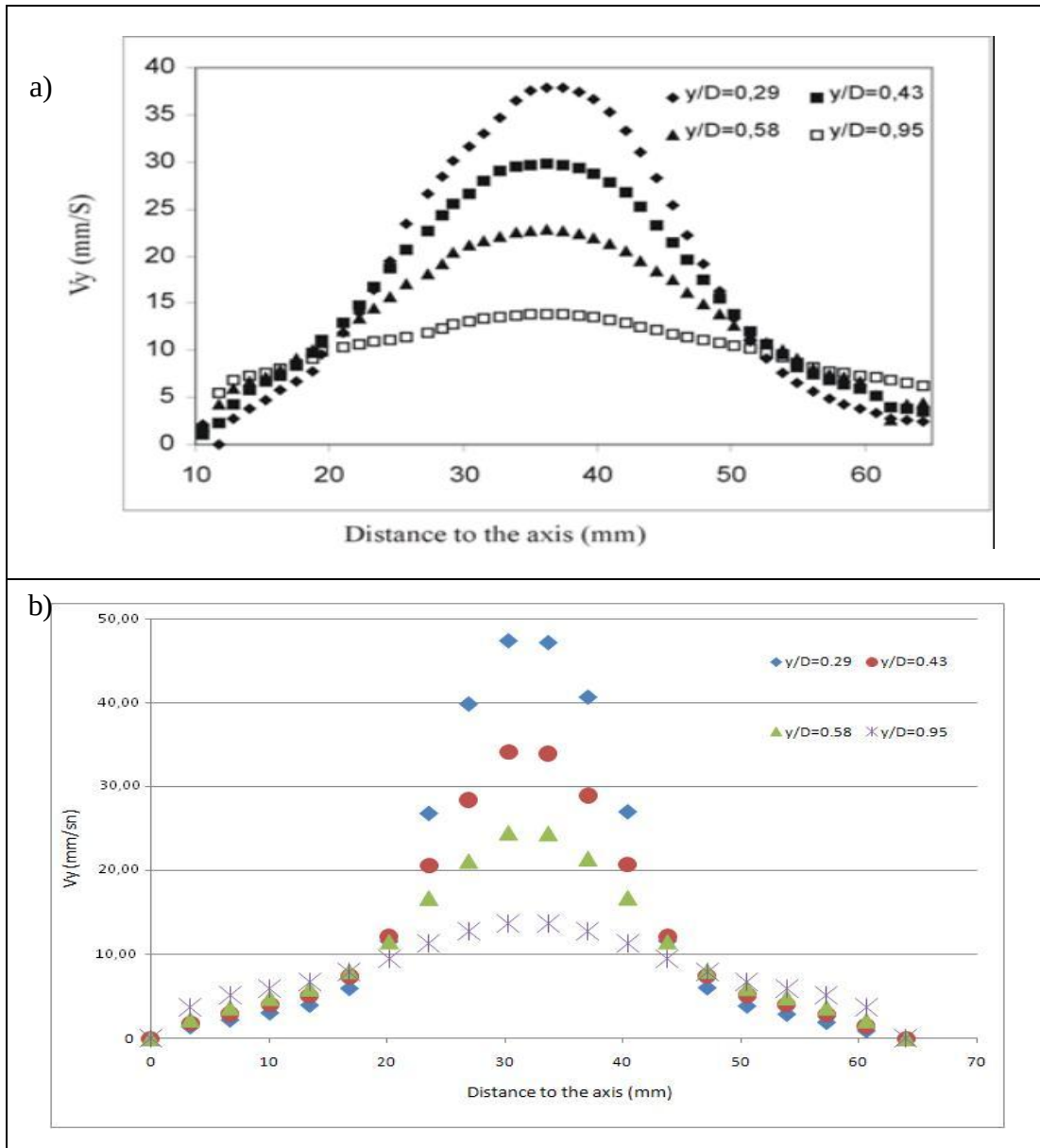


Figure 4.35. a) Experimental result for axial velocity profiles V_y at $Re = 1989$
 b) Numerical simulation result for axial velocity profiles V_y at $Re = 1989$

Same as in V_x profiles, from Reynolds number of 233 through 1989 for V_y velocities we also observed the same profiles. Difference is the effect of the Reynolds number on the maximum V_y value.

Up to now we compared the results of experimental study of Fakilar, M.S., (2004) with the results of numerical simulation, and we prove that numerical simulation properly works. Now we can go one step further by comparing the numerical simulation done for water with numerical simulation done for aqueous solution of polyacrylamide by using FLUENT.

4.2. Comparison of Water and Polyacrylamide Numerical Solutions Results

Polyacrylamide is a brittle, white, odorless polymer, which can be manufactured by free radical polymerization of acrylamide polymer. It is versatile and synthetic, and is therefore, used worldwide to improve commercial products and processes. It is water soluble at all concentrations, temperatures and pH values. Since most of the uses of polyacrylamide involve aqueous systems, the physical properties of aqueous solutions are important.

Viscosity: The viscosity of polyacrylamide in water increases dramatically with molecular weight while it decreases with increasing temperature.

Shear sensitivity: At relatively low shear rates, aqueous solutions of polyacrylamide are shear thinning. However, at high shear rates rupturing the polymer chains can degrade the molecular weight.

In this study, the parameters of aqueous polyacrylamide are defined to Polyflow software according to C_{04} concentration given in experimental study of BROBOANA et al. (2007).

Material parameters	Concentration C_{01}	Concentration C_{02}	Concentration C_{03}	Concentration C_{04}
η^* [Pas]	0.0007	0.0016	0.0023	0.0028
k [s]	0.00009	0.00018	0.0002	0.00018
n [-]	0.0001	0.00137	0.1936	0.21

Also Cross law model was chosen from Polyflow simulation software for the simulation of C_{04} concentration.

Cross law formula is given as;

$$\eta(\dot{\gamma}) = \frac{\eta_0}{1 + (\lambda\dot{\gamma})^{n-1}}$$

where $k = \lambda$ as compared with the parameter in table.

Numerical simulations for two different fluids are compared from Figure 4.36 to Figure 4.49. Figure numbers with extension letter “a” (like in Fig. 4.36a) show the results for water while those with extension letter “b” (like in Fig. 4.36b) show the results for polyacrylamide solution. Newtonian and non-Newtonian results are compared only for $Re=233$ and $Re=336$. Results for smaller Reynolds numbers, $Re=1$, 10 and 100 are also given for polyacrylamide.

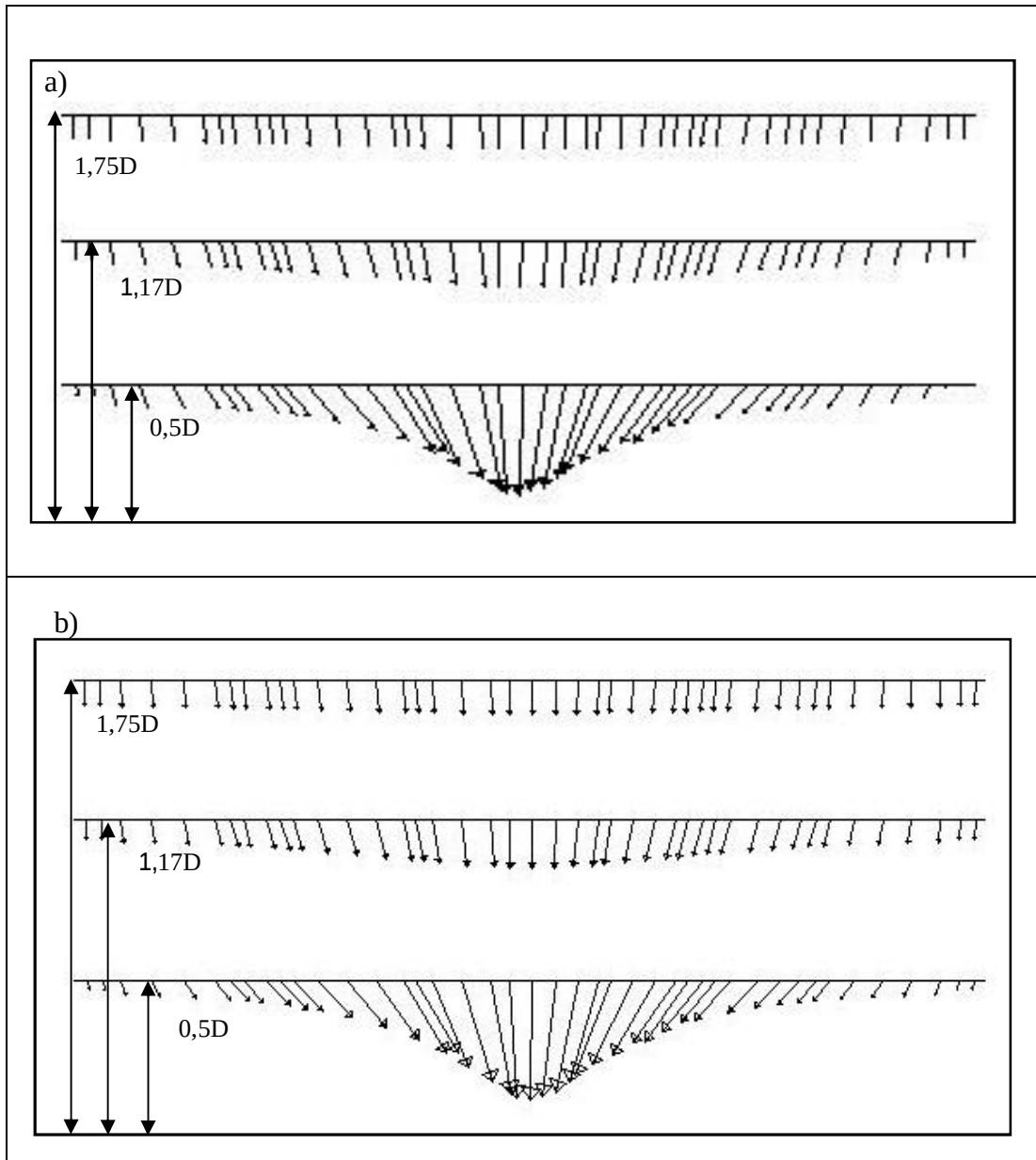


Figure 4.36.a) Average velocity vectors of numerical solution for water at three different locations ($0.5D$, $1.17D$, $1.75D$) for $Re=233$

b) Average velocity vectors of numerical solution for polyacrylamide at three different location ($0.5D$, $1.17D$, $1.75D$) for $Re=233$

In Figures 4.36a and 4.36b we can see same shapes and directions of velocity vectors at same distance above the contraction plane. Although the Reynolds numbers are same for both figures, the magnitudes of the vectors at same distance of each figures are different because of the difference in viscosity of fluids.

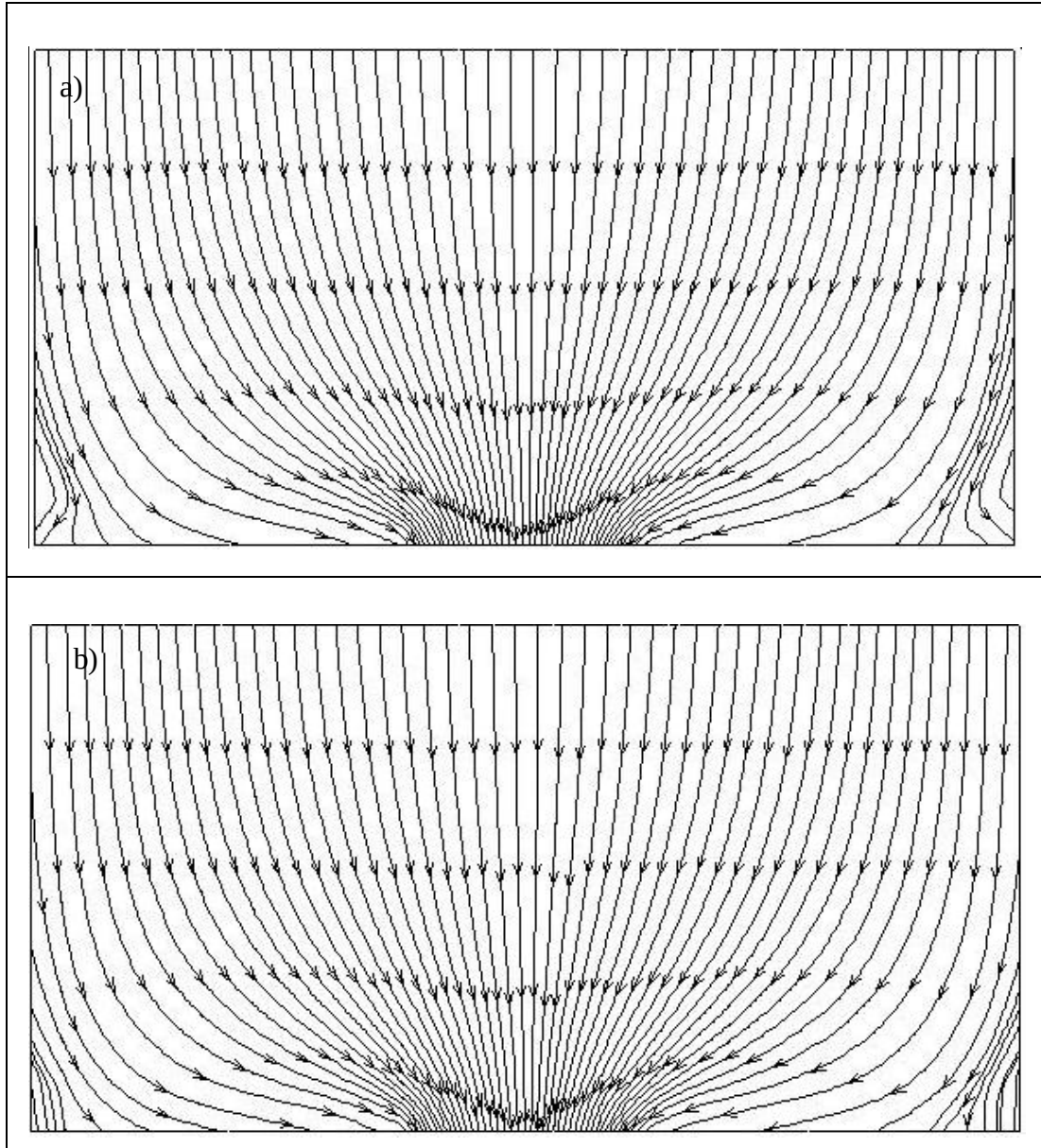


Figure 4.37. a) Streamlines of numerical simulation for water at $Re=233$
b) Streamlines of numerical simulation for polyacrylamide at $Re=233$

When we compare the Figures 4.37a and 4.37b, we can see the difference for the sizes of vorticities between the streamlines for two different fluids.

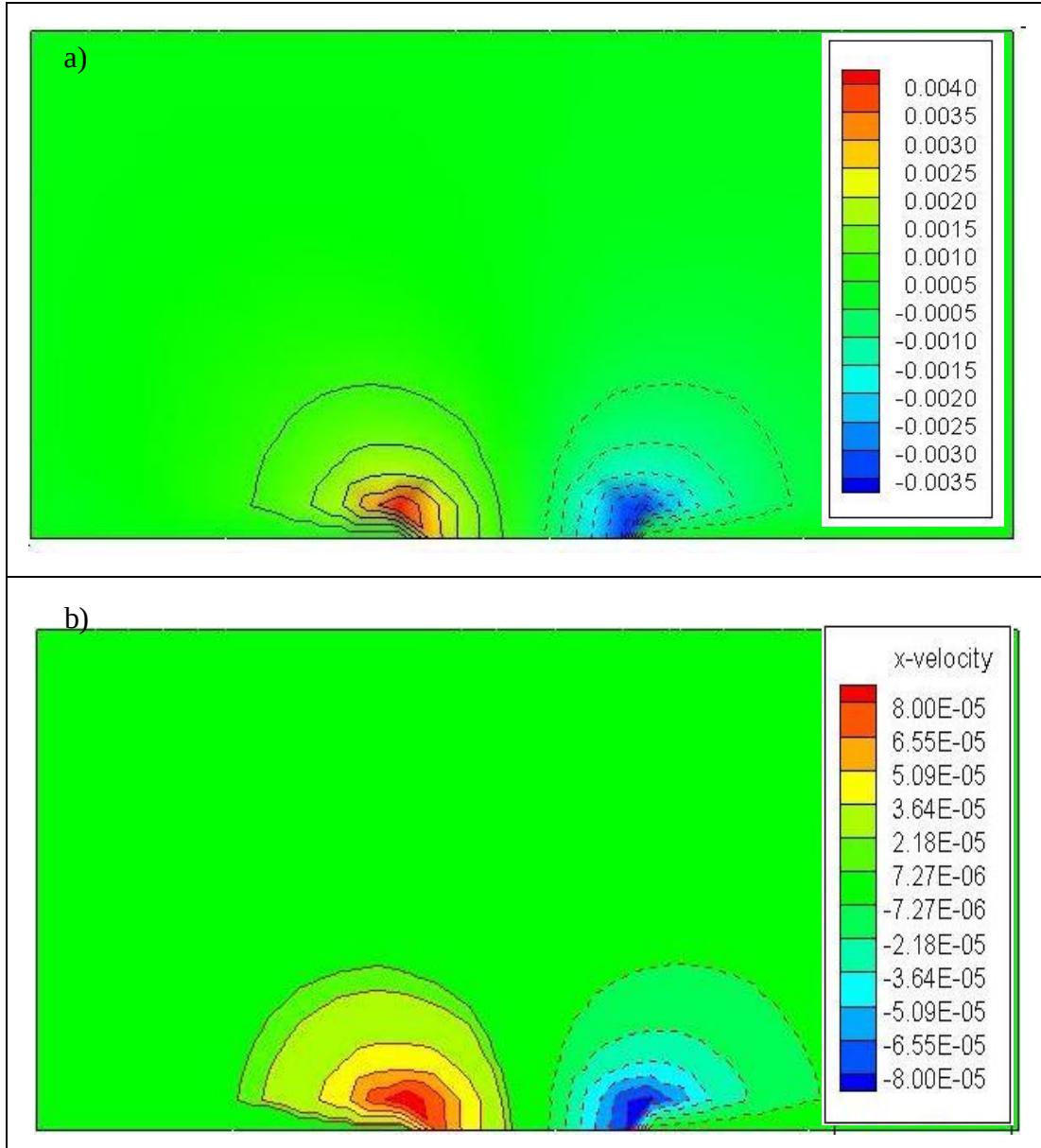


Figure 4.38.a) Average u velocity contour of numerical solution for water at $Re=233$
 b) Average u velocity contour of numerical solution for polyacrylamide at $Re=233$

We can see the same shapes of u velocity contours from the Figures 4.38a and 4.38b, difference is the values of velocities due to the different viscosity of fluids. We see that the x-velocity of non-Newtonian fluid is much smaller than the Newtonian fluid.

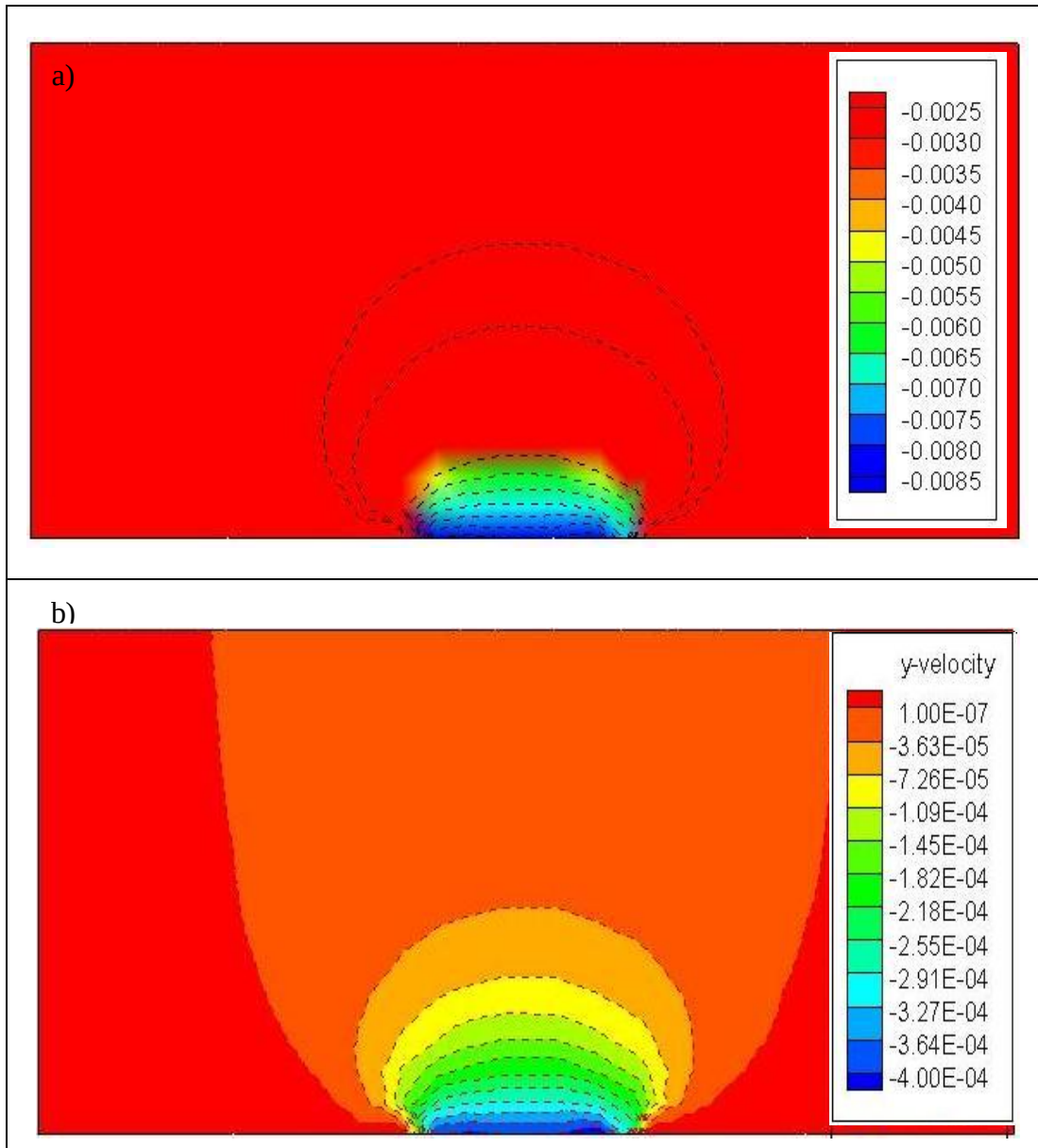


Figure 4.39. a) Average v velocity contour of numerical solution for water at $Re=233$
 b) Average v velocity contour of numerical solution for polyacrylamide at $Re=233$

Same as in comparison of the u velocity contours, v velocity contours are similar and values of velocities also different due to the viscosity difference as we can see from Figures 4.39a and 4.39b. Again, y-velocity of non-Newtonian fluid is much smaller than the Newtonian fluid.

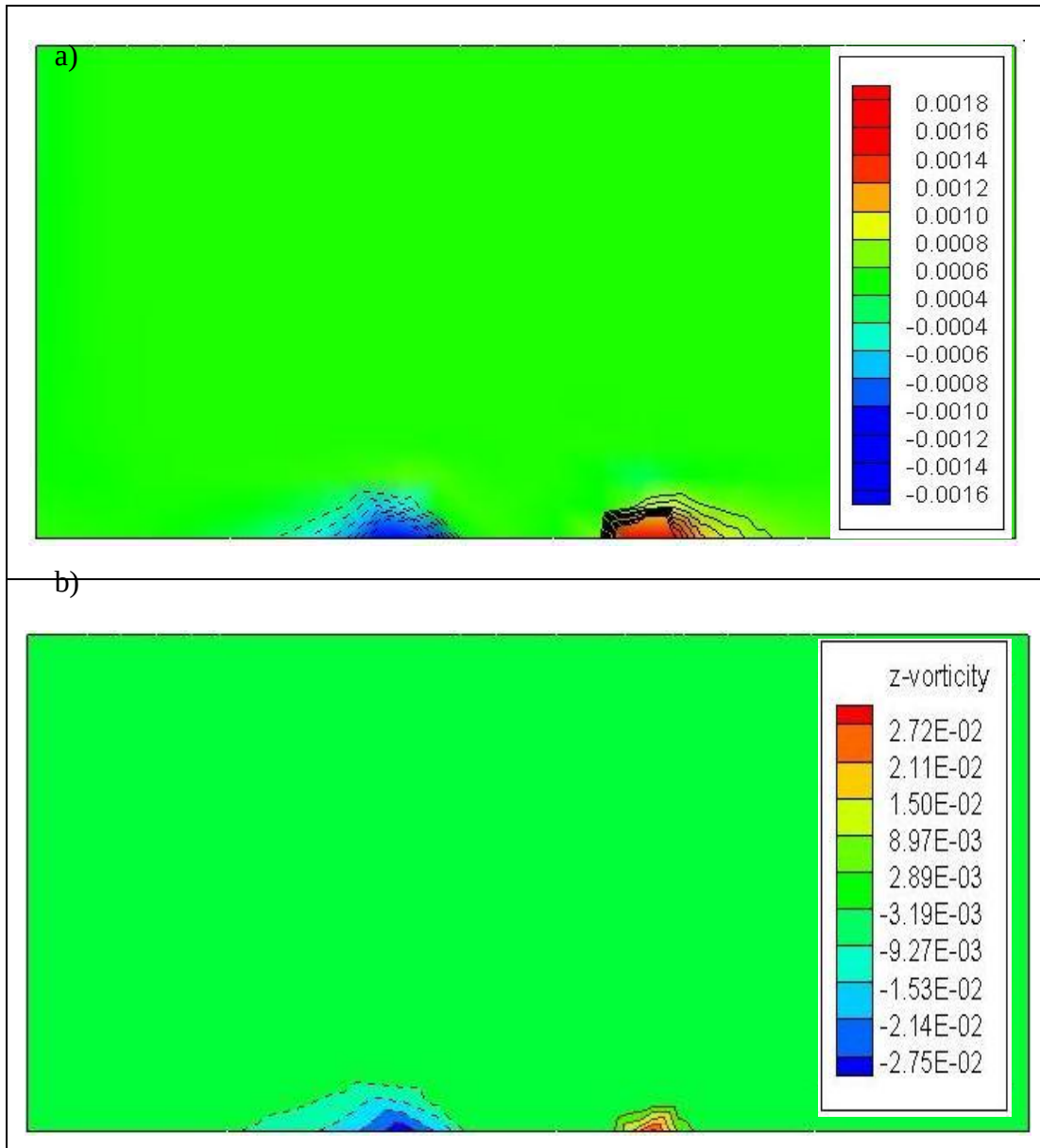


Figure 4.40.a) Average w vorticity contour of numerical solution for water at $Re=233$

b) Average w vorticity contour of numerical solution for polyacrylamide at $Re=233$

As a result of comparison between the vorticity contours of two different fluids for Reynolds numbers of 233, difference that we observed is distance between the positive and negative contours. Vorticity values for non-Newtonian fluid are higher than Newtonian fluid.

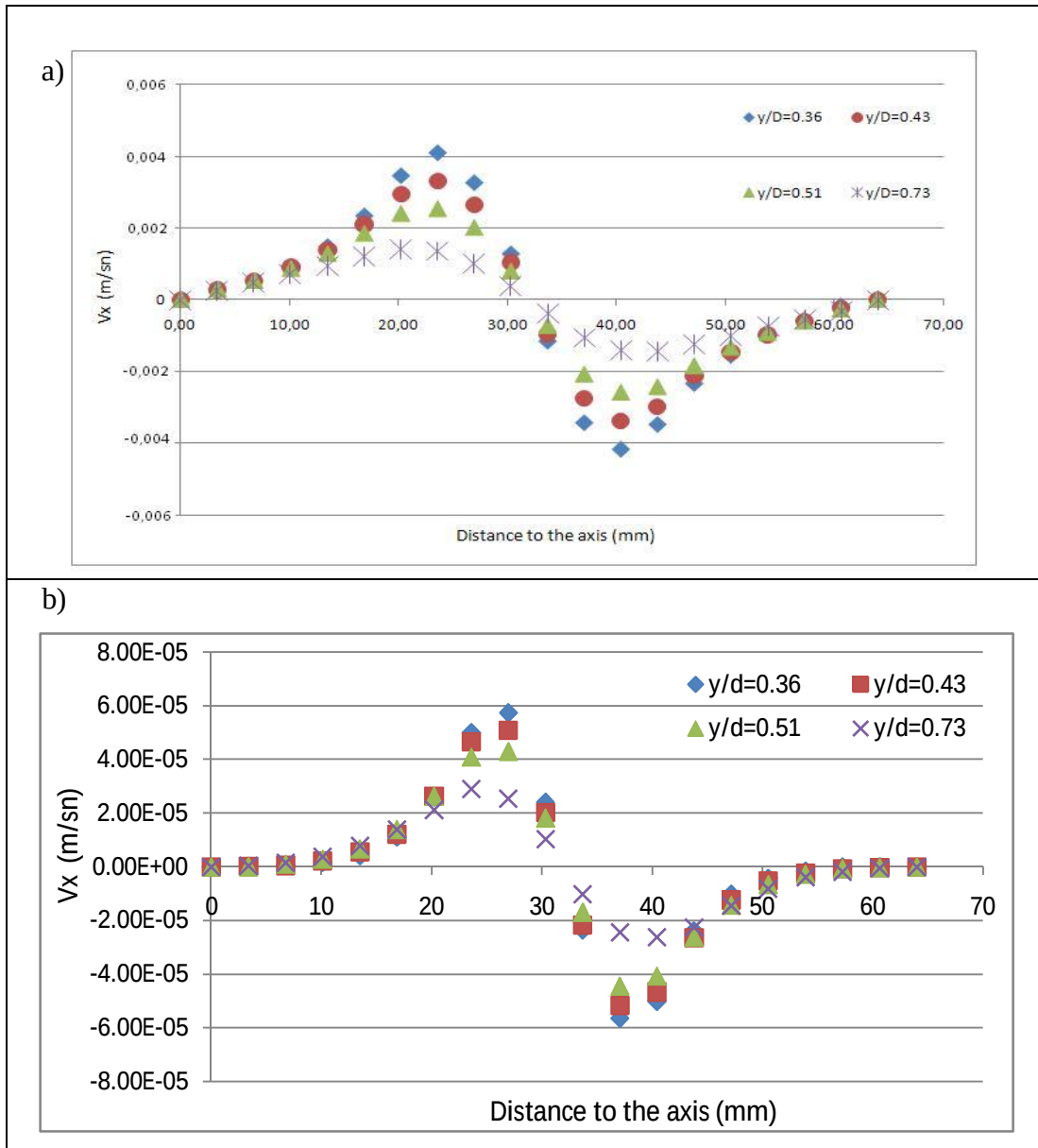


Figure 4.41.a) Average V_x velocity profiles for different y/D ratios of numerical solutions for water at $Re=233$

b) Average V_x velocity profiles for different y/D ratios of numerical solutions for polyacrylamide at $Re=233$

The V_x profiles for two different fluids are much more similar when compared with the experimental and numerical simulation similarities. Again the difference is the maximum and minimum values. V_x velocities of non-Newtonian fluid is much smaller than Newtonian fluid. Also, near the tube walls, non-

Newtonian fluid velocities are almost zero meaning a much more still fluid compared to water.

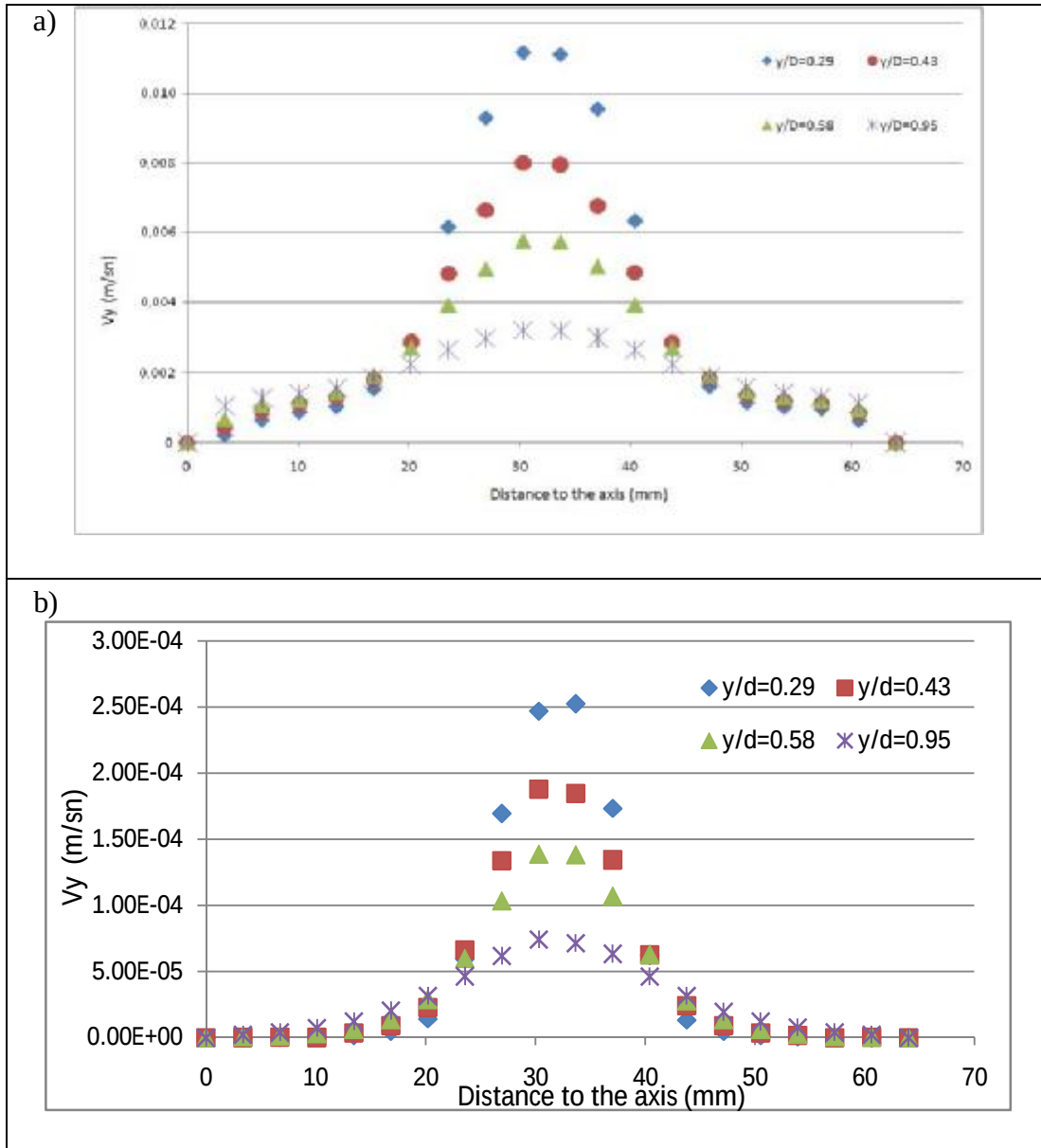


Figure 4.42.a) Average V_y velocity profiles for different y/D ratios of numerical solutions for water at $Re=233$

b) Average V_y velocity profiles for different y/D ratios of numerical solutions for polyacrylamide at $Re=233$

Same as in comparison of V_x profiles for Reynolds number of 233, results for V_y profiles of two different fluids are much more similar when compared with the

similarity of experimental and numerical simulation. Again, non-Newtonian fluid velocities are much smaller than Newtonian fluid. Away from contraction, non-Newtonian fluid velocities in y direction are almost zero, which is a main difference compared to Newtonian fluids.

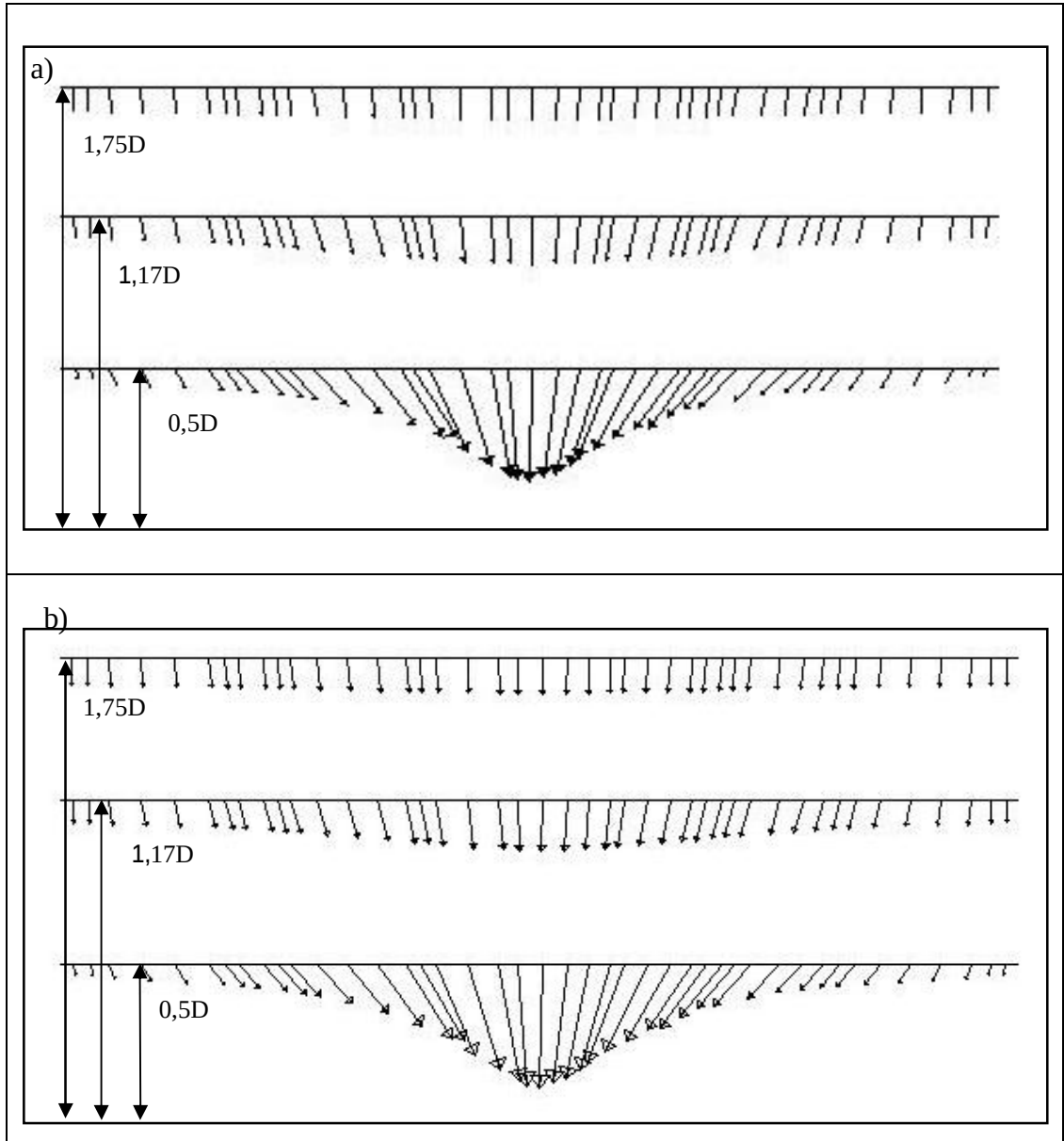


Figure 4.43.a) Average velocity vectors of numerical solution for water at three different locations (0.5D, 1.17D, 1.75D) for $Re=336$
 b) Average velocity vectors of numerical solution for polyacrylamid at three different location (0.5D, 1.17D, 1.75D) for $Re=336$

Figures 4.43a and 4.43b shows the similarities of both fluids in shape and direction. Difference is the magnitudes due to the viscosity.

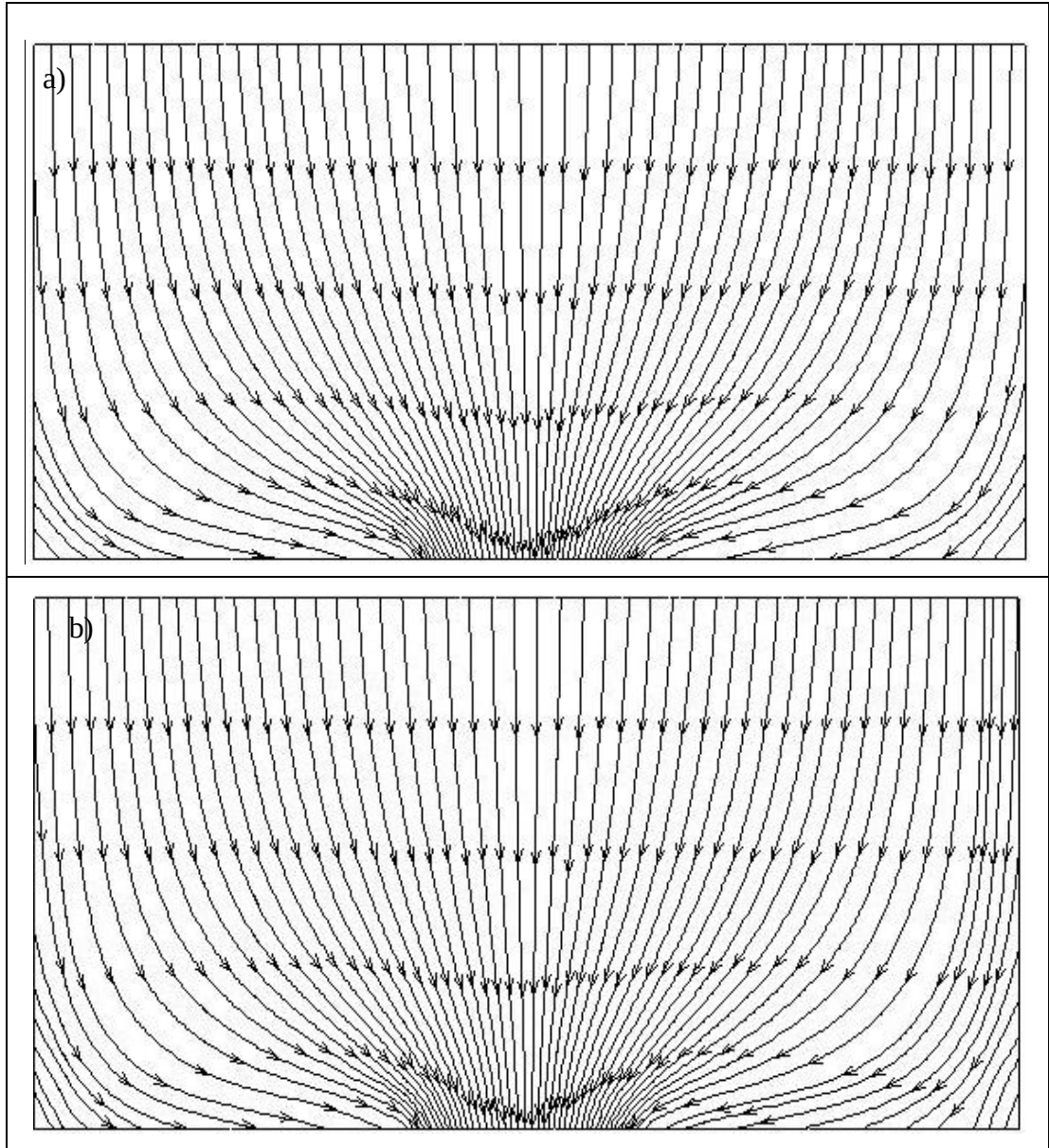


Figure 4.44. a) Streamlines of numerical simulation for water at $Re=336$
b) Streamlines of numerical simulation for polyacrylamide at $Re=336$

In Figure 4.44a we can see that the vortex which we observed for Reynolds number of 233 disappears when flow reaches the Reynolds number of 336. In Figure 4.44b, vorticity observed for Reynolds number of 233 is also disappears.

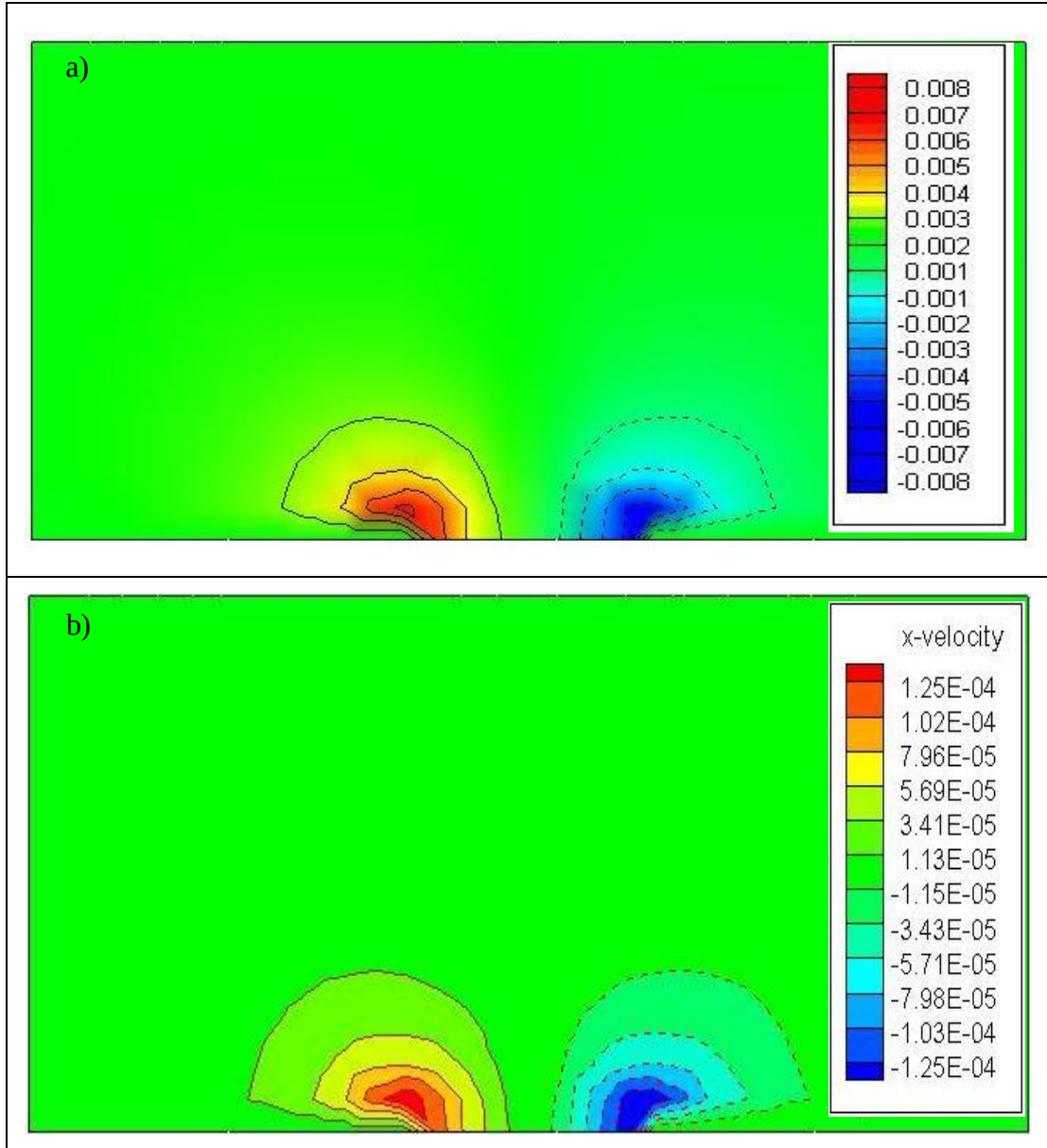


Figure 4.45. a) Average u velocity contour of numerical solution for water at $Re=336$
 b) Average u velocity contour of numerical solution for polyacrylamide at $Re=336$

When we compare the contours in Figures 4.45a and 4.45b again we can see the similar shapes. Also we observed that the velocity values of both contours are increased according to increase of Reynolds number.

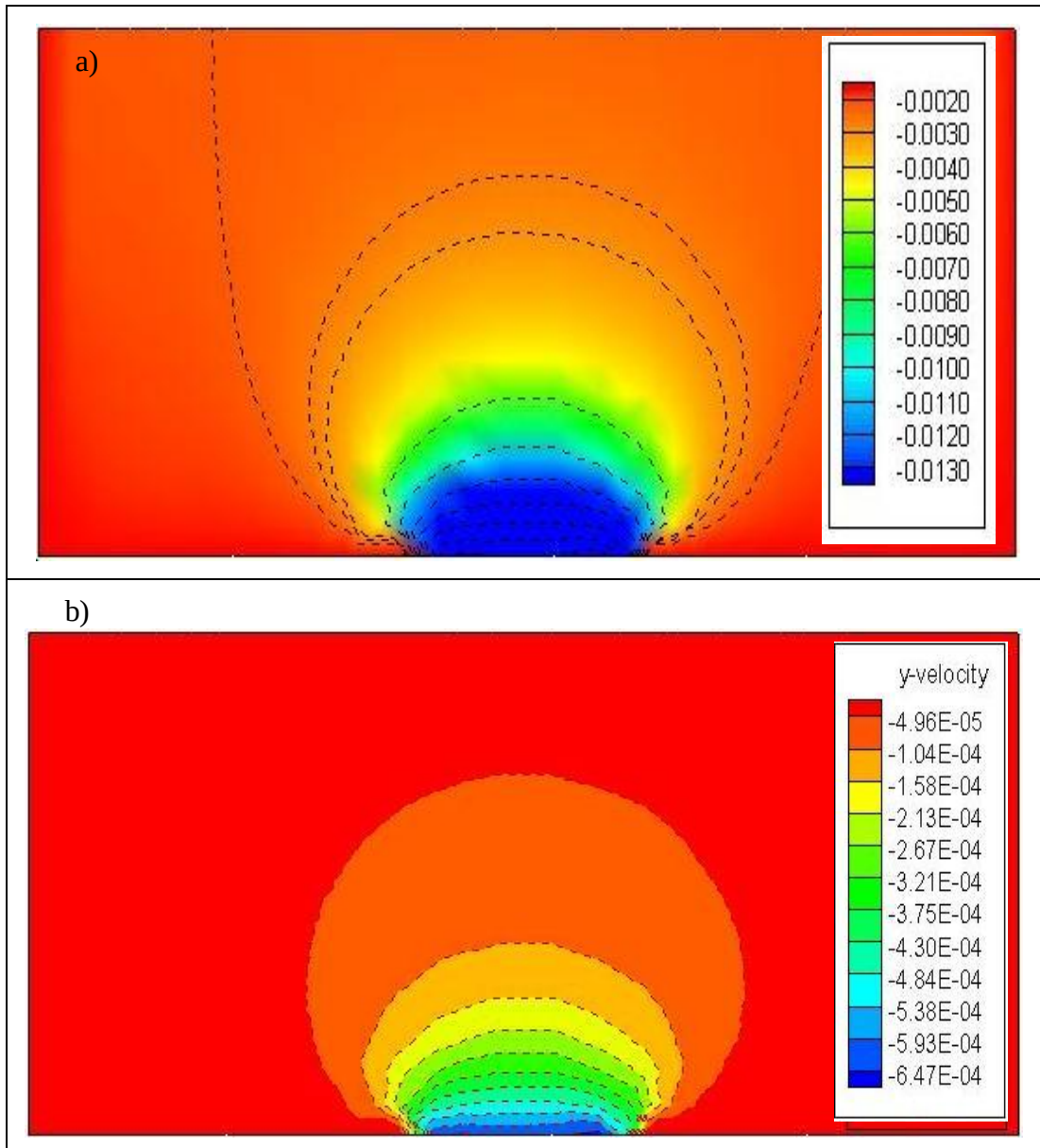


Figure 4.46. a) Average v velocity contour of numerical solution for water at $Re=336$
 b) Average v velocity contour of numerical solution for polyacrylamide at $Re=336$

Figures 4.46a and 4.46b shows the numerical simulation results of v velocity for Reynolds number of 336. We can see that the figures are very similar and values of v velocities are increased due to the increase of Reynolds number from 233 to 336.

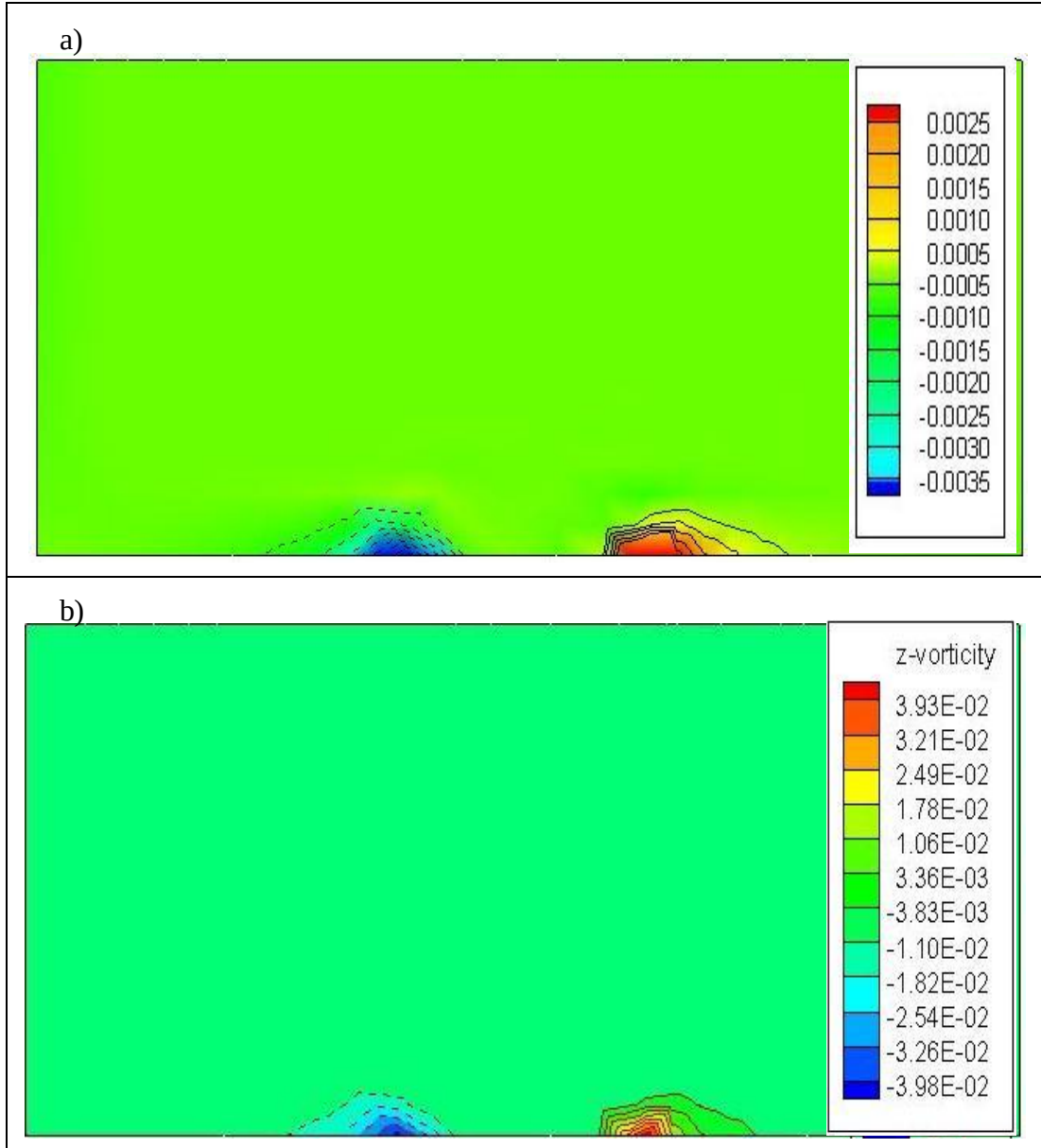


Figure 4.47.a) Average w vorticity contour of numerical solution for water at $Re=336$

b) Average w vorticity contour of numerical solution for polyacrylamide at $Re=336$

Figures 4.47a and 4.47b shows the numerical simulation results for w vorticity of water and aqueous solution of polyacrylamide respectively. As can be seen shapes of contours are similar and also when we compare Figures 4.40 and 4.47, we can see the higher values of vorticity in Figure 4.47 because of higher Reynolds number.

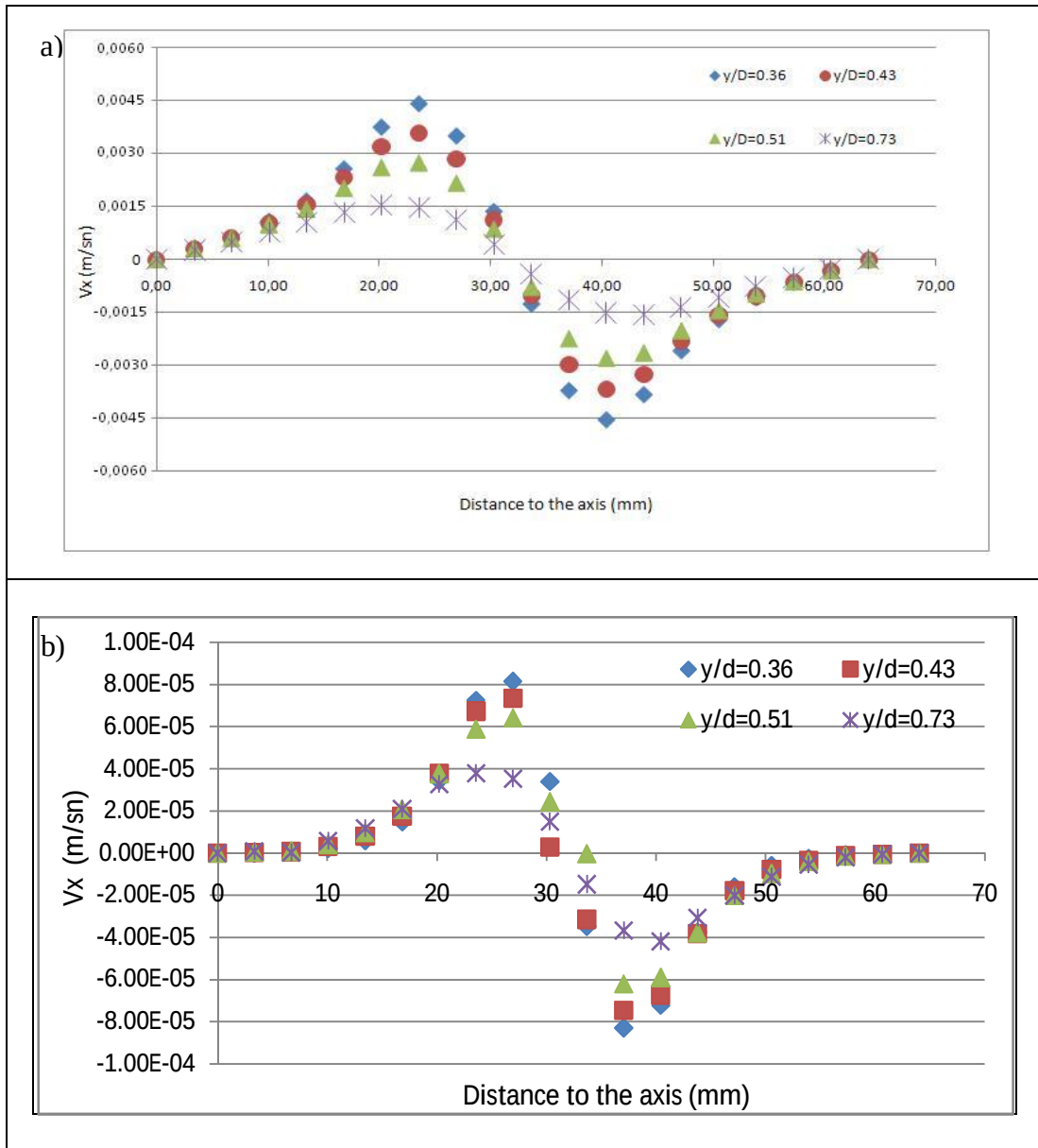


Figure 4.48.a) Average V_x velocity profiles for different y/D ratios of numerical solutions for water at $Re=336$

b) Average V_x velocity profiles for different y/D ratios of numerical solutions for polyacrylamide at $Re=336$

As can be seen from the Figures above, the V_x velocity profiles of both numerical simulations for Reynolds number of 336 are very similar.

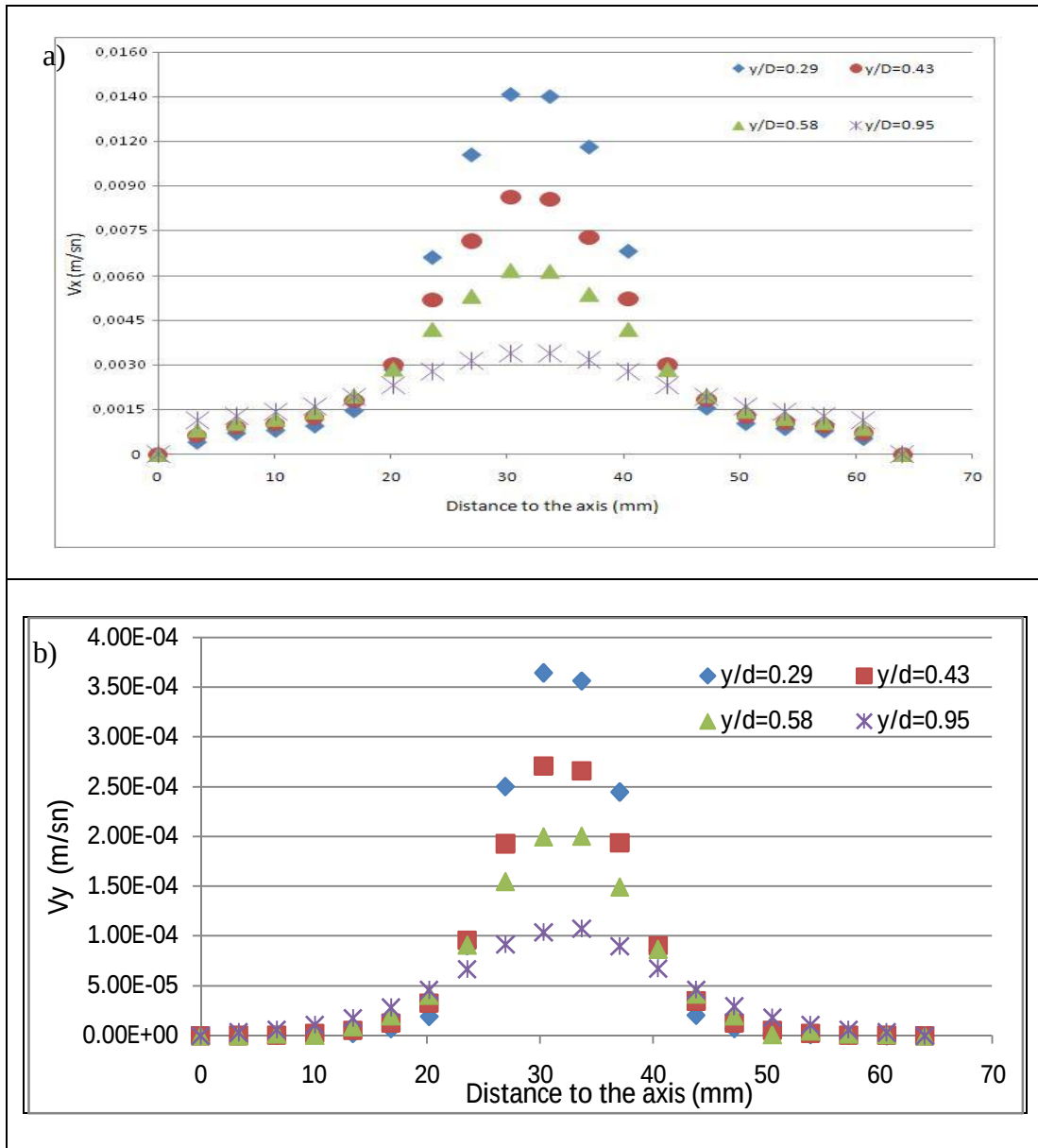


Figure 4.49.a) Average V_y velocity profiles for different y/D ratios of numerical solutions for water at $Re=336$

b) Average V_y velocity profiles for different y/D ratios of numerical solutions for polyacrylamide at $Re=336$

Same as in other comparisons of two different numerical simulations, Figures above seems very similar.

4.3. Results of Numerical Simulation for aqueous polyacrylamide solution

Numerical simulation results of polyacrylamide for Reynolds number of 1,10 and 100 are shown in Figure 4.50 through Figure 4.58. Although the fluid shows the same characteristics in these Reynolds numbers also, the difference is the numerical values due to the difference of the viscosity of fluid which is used in these simulations. As can be seen from Figures 4.51, 4.54 and 4.57 when the Reynolds number increases the value of u,v velocities and w vorticity also increases linearly. V_x profile shows the same shape as in other examples, positive on left and negative on right side of the contraction axis. V_y profile also shows the same shape with other examples.

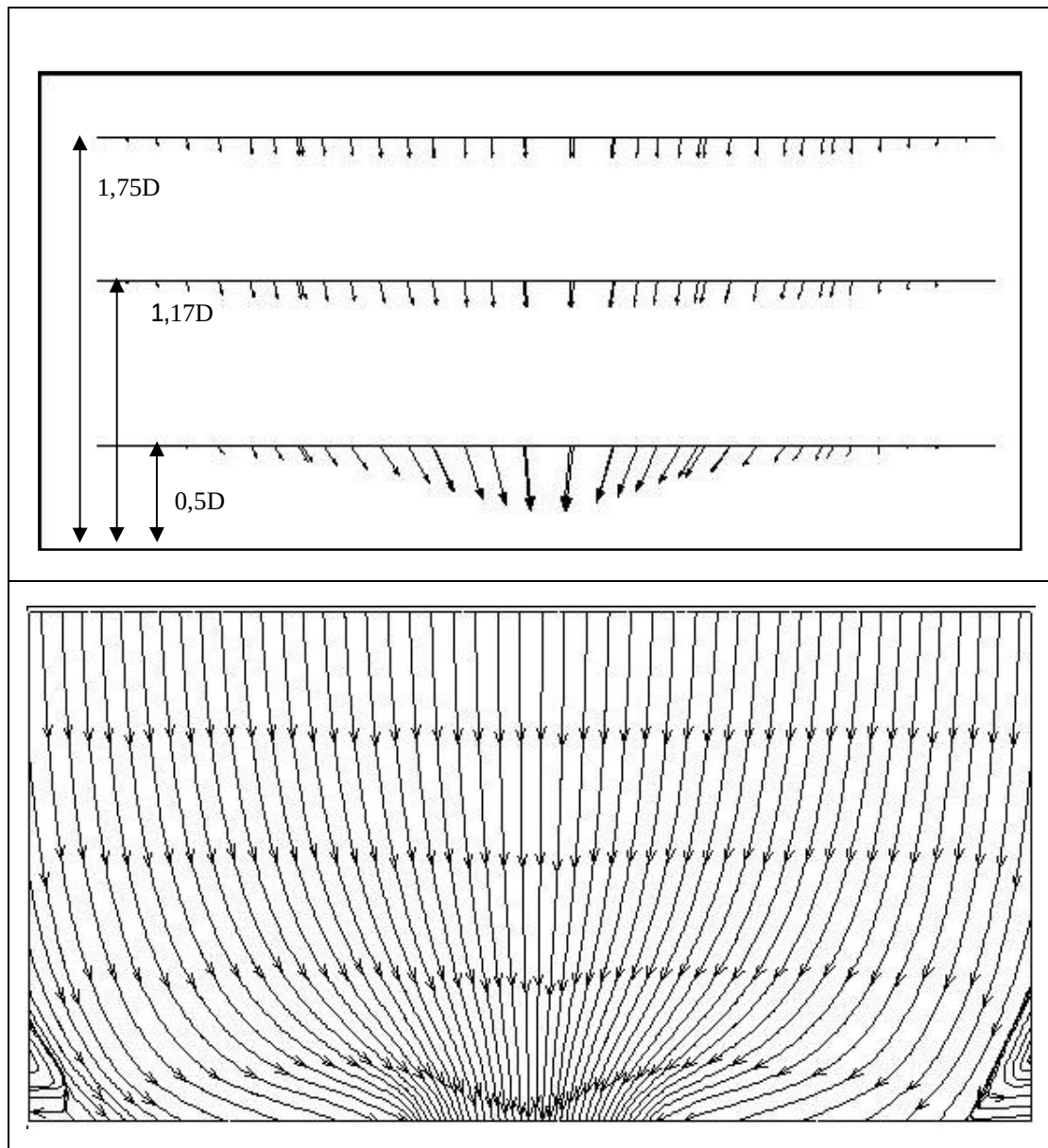


Figure 4.50. Velocity vectors and streamlines of numerical Simulation with FLUENT for $Re=1$

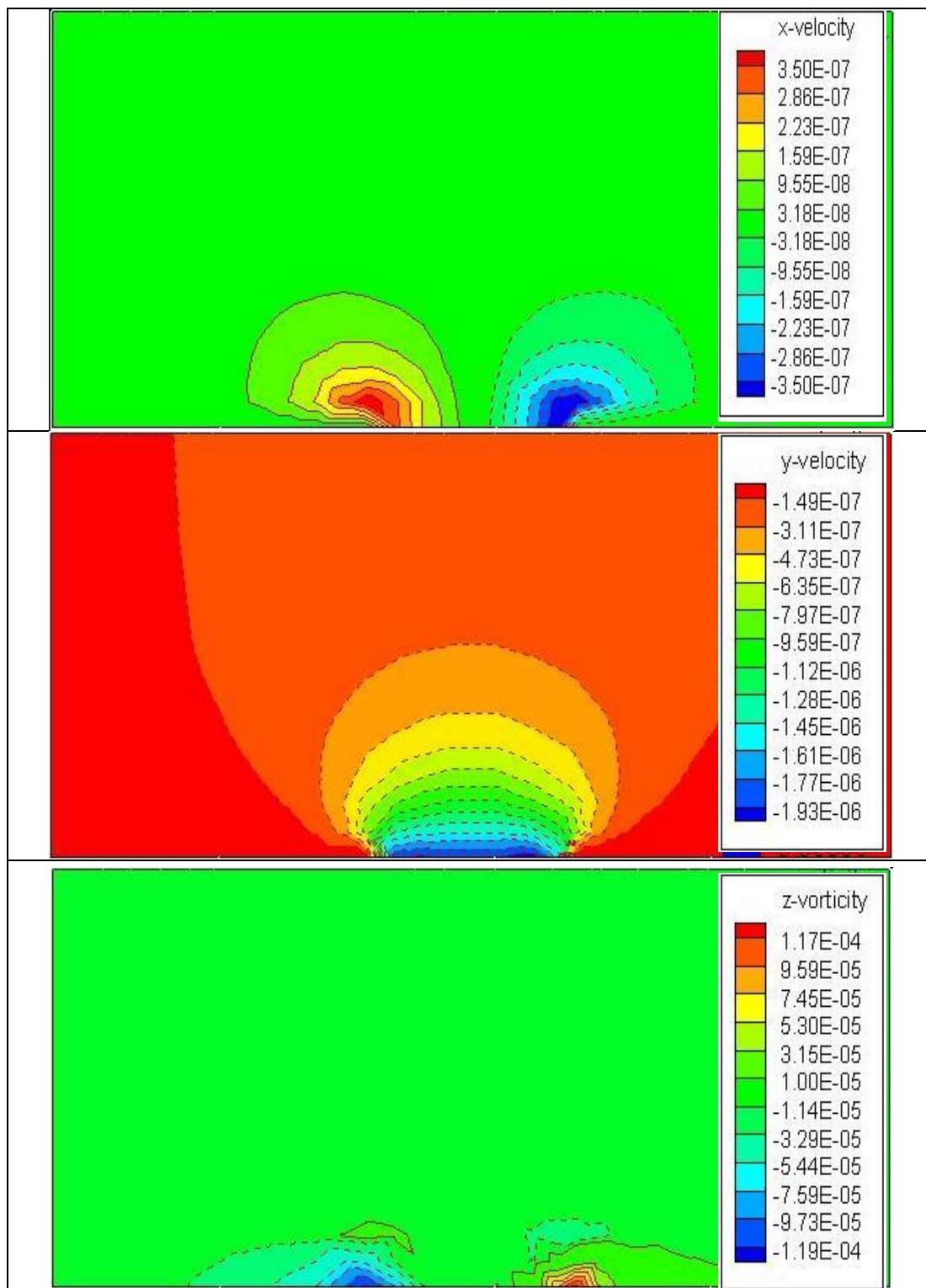


Figure 4.51. Velocity contours and vorticity contours of numerical Simulation with FLUENT for $Re=1$

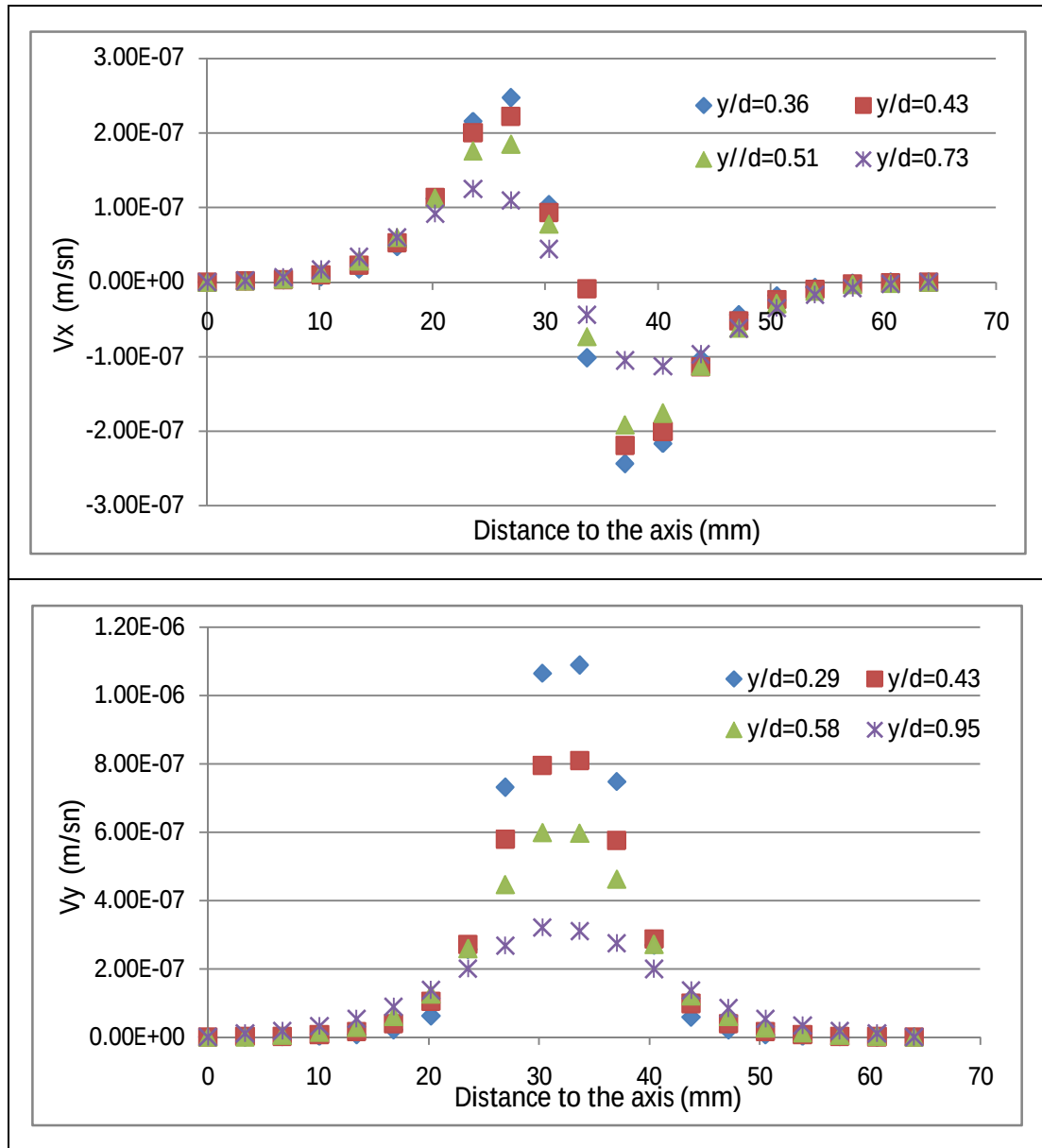


Figure 4.52. Average V_x and V_y profiles of numerical simulation with FLUENT for $Re=1$

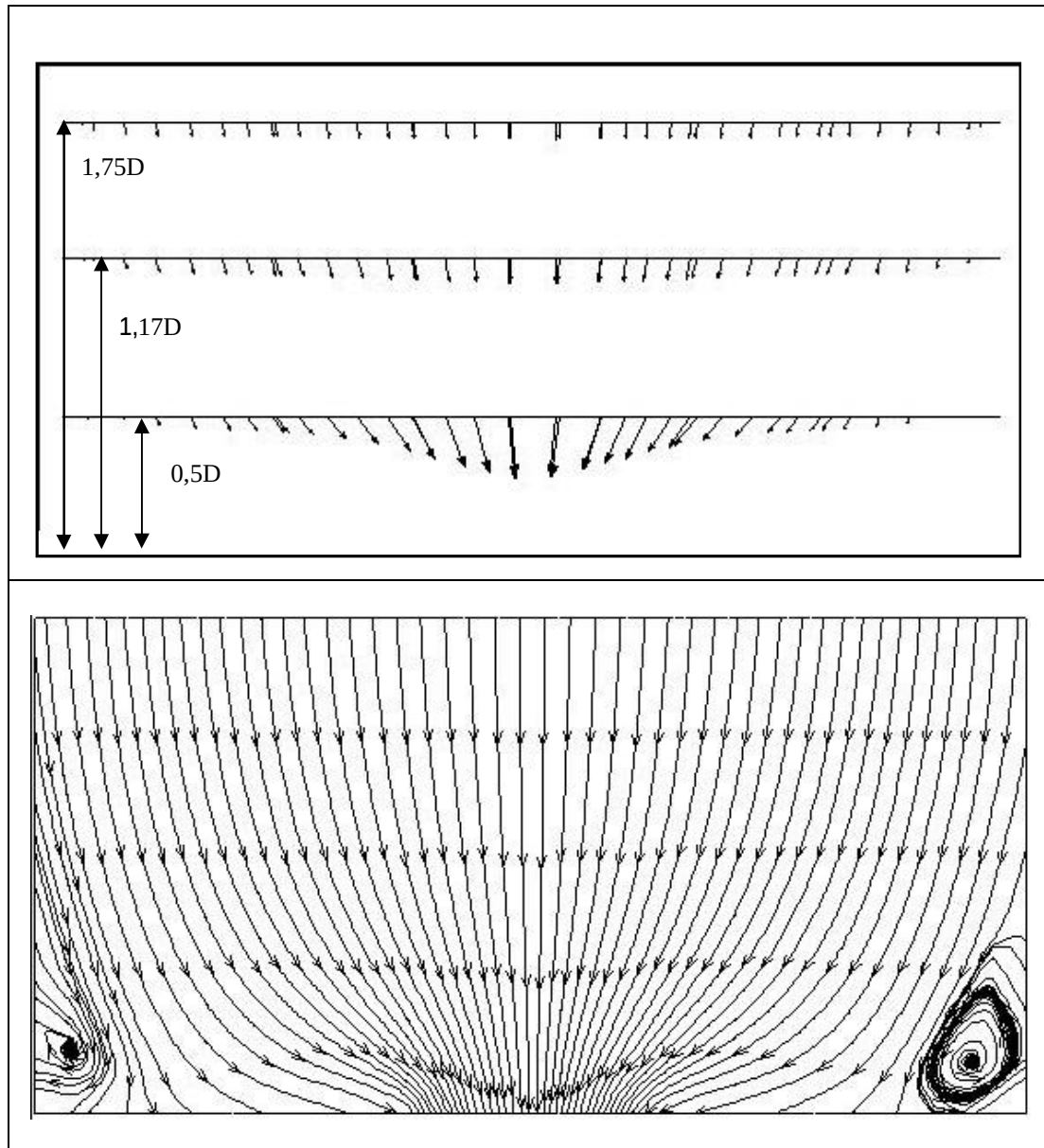


Figure 4.53. Velocity vectors and streamlines of numerical simulation with FLUENT for $Re=10$

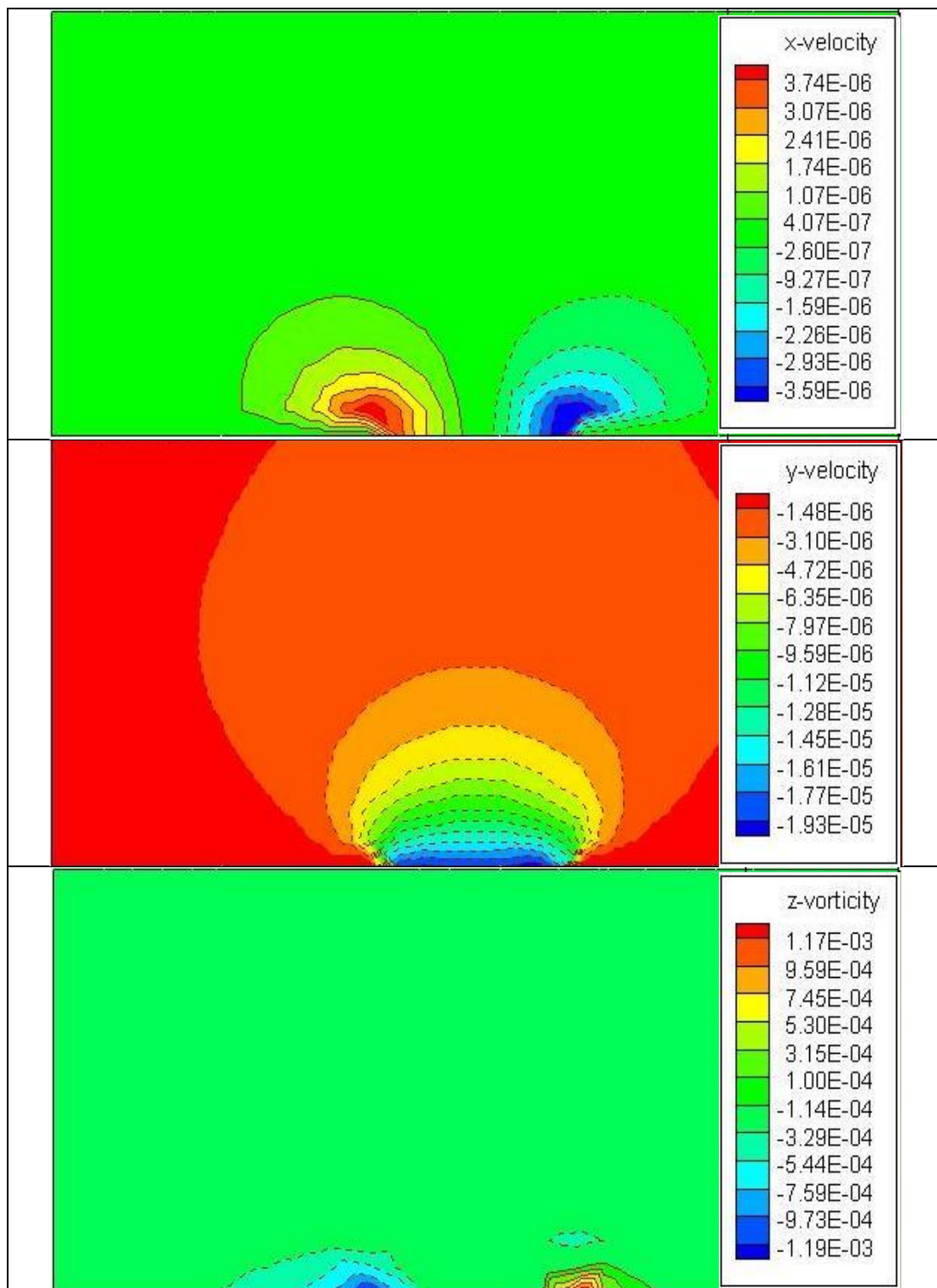


Figure 4.54. Velocity contours and vorticity contours of numerical simulation with FLUENT for $Re=10$

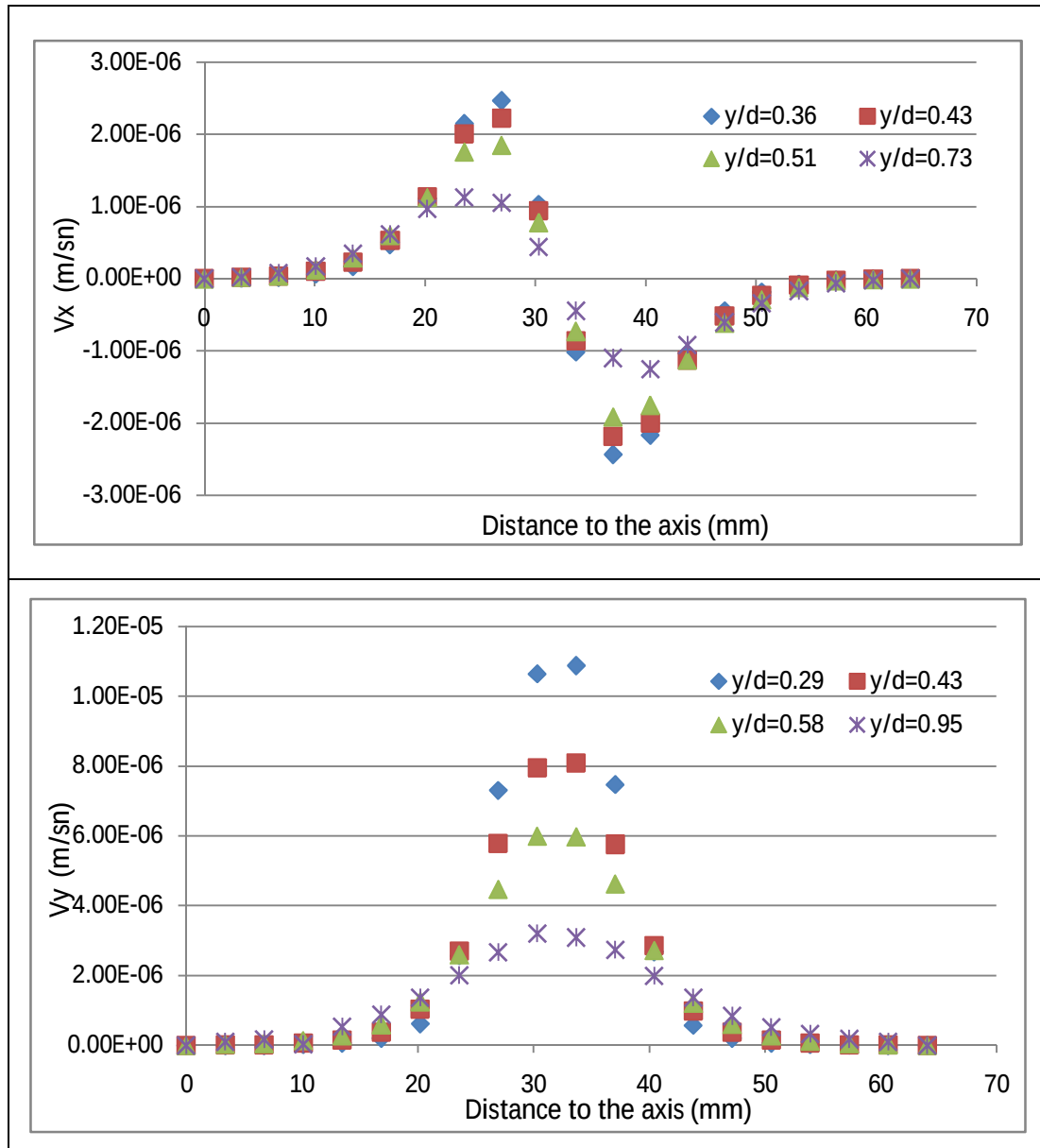


Figure 4.55. Average V_x and V_y profiles of numerical simulation with FLUENT for $Re=10$

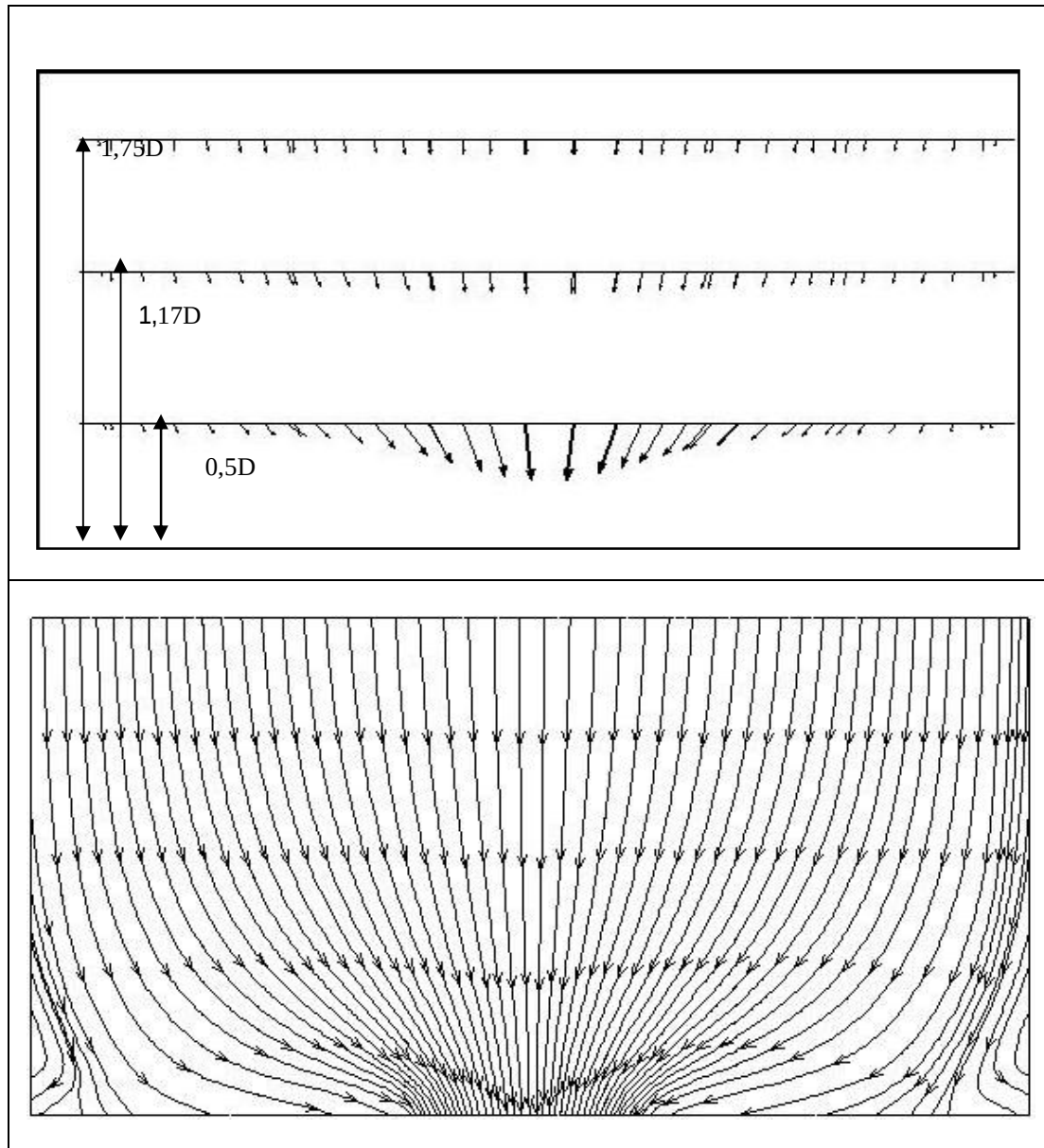


Figure 4.56. Velocity vectors and streamlines of numerical simulation with FLUENT for $Re = 100$

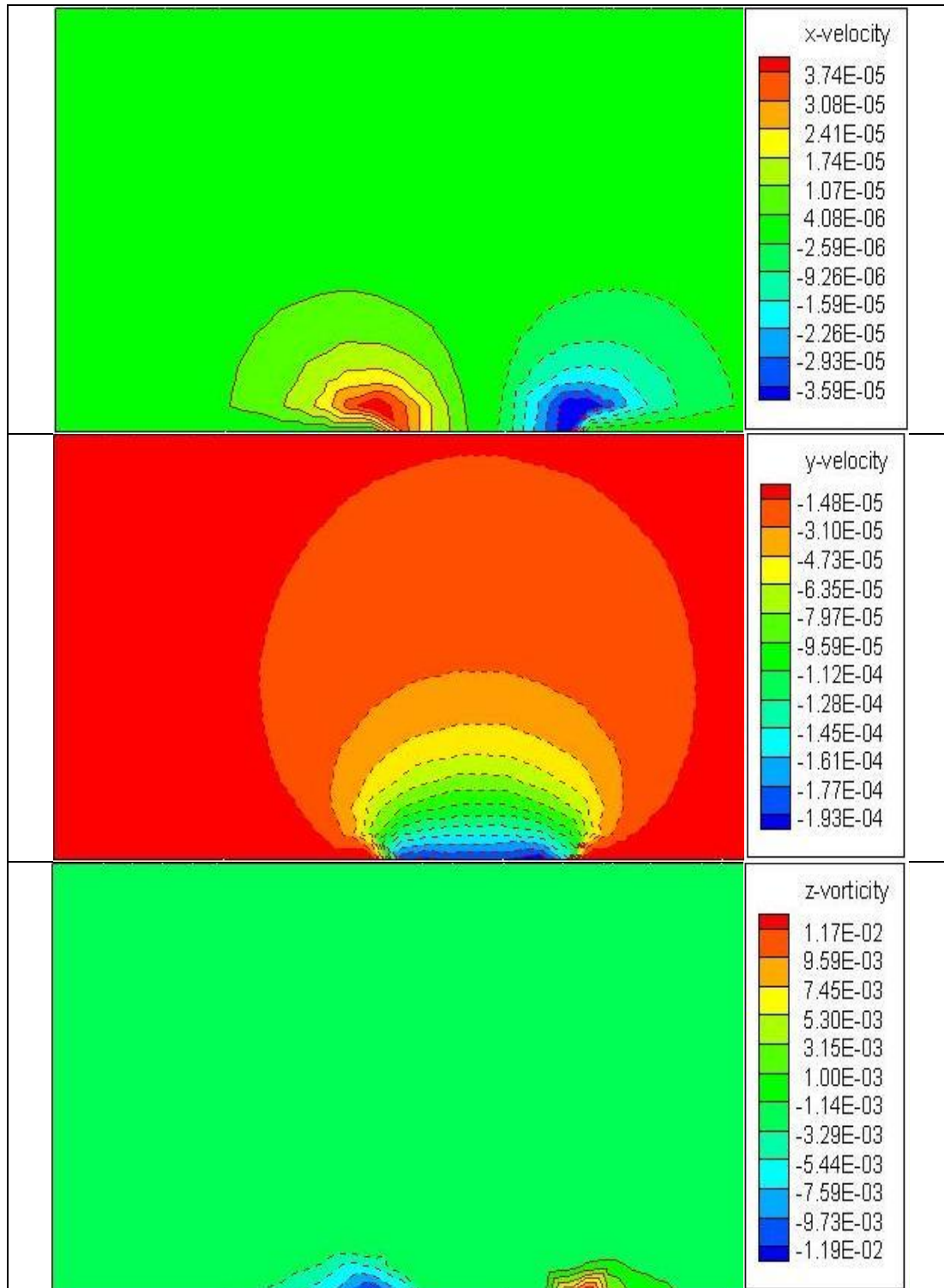


Figure 4.57. Velocity contours and vorticity contours of numerical simulation with FLUENT for $Re=100$

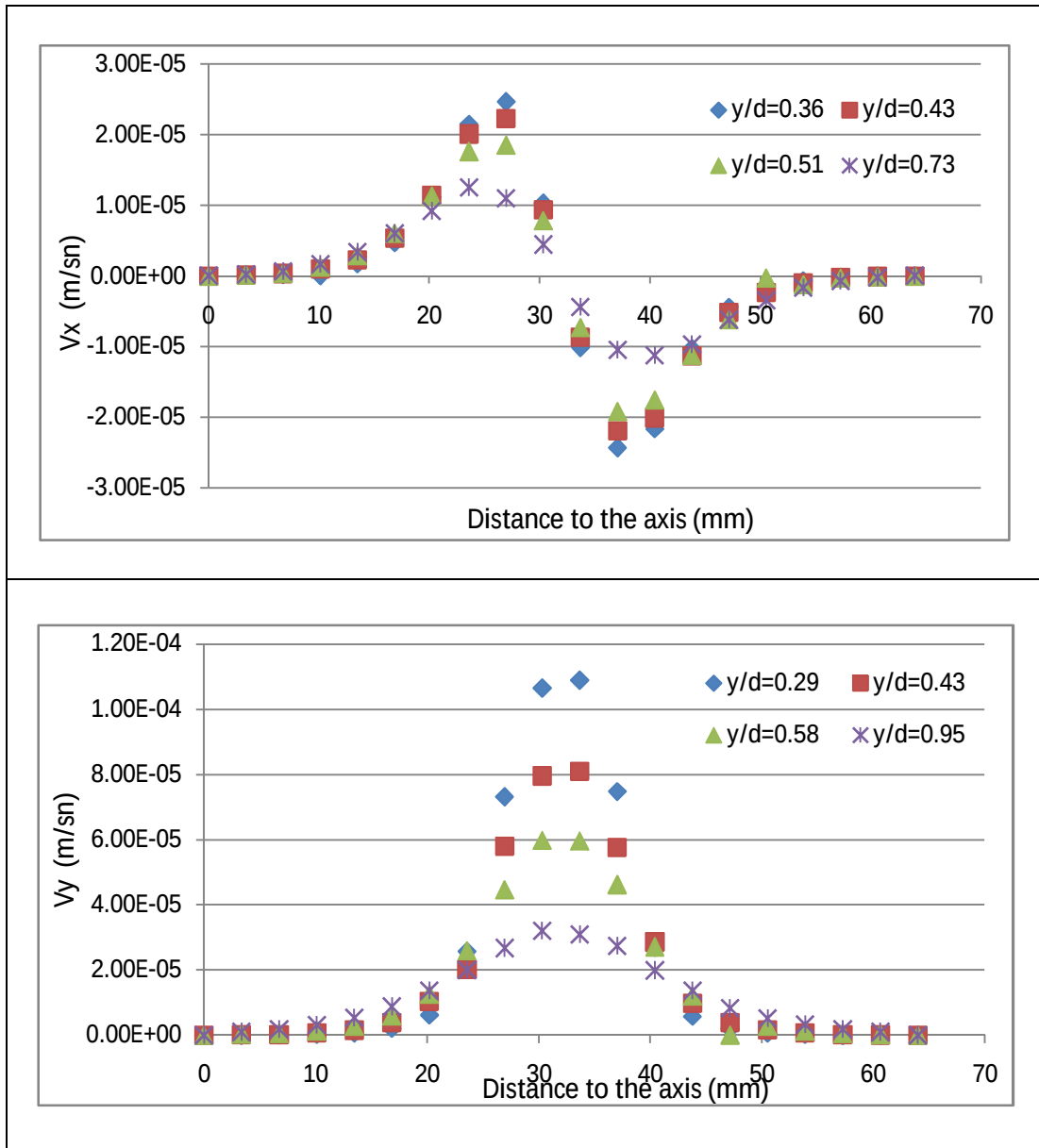


Figure 4.58. Average V_x and V_y profiles of numerical simulation with FLUENT for $Re=100$

5.CONCLUSION

Flow analysis of a non-Newtonian fluid is one of the most popular subjects of researchers over the past five decades. At the beginning researchers were dealing with the measurements of the properties of non-Newtonian fluids, now the researches focus on analyzing of flow with a numerical simulation. Because the polymer melts are widely used in industrial applications, polymer solutions are the most popular fluid of research subjects. One of the most widely studied entry flow geometry is circular sudden contraction flow which is the most realistic case as compared with the industrial applications.

In the first phase of, the flow of a Newtonian fluid through a sudden contraction pipe is analyzed with Numerical simulation software and results are compared with the results of an experimental study of Fakilar, M.S. (2004).

General characteristics of the flows in numerical simulations are similar to results of the experimental study of Fakilar, M.S. (2004). As we increase the Reynolds number, velocity contours of u , v and vorticity contour of w gets larger. For the Reynolds number of 233, we observe a weak vortex formation but as the Reynolds number reaches the value of 336 vortex formation disappears.

In the second phase of this thesis, numerical simulation results for water and polyacrylamide solution were compared to see the effect of inelasticity of the fluid. It was seen that u and v velocities of non-Newtonian fluid are much smaller than Newtonian fluid. However, vorticity values for non-Newtonian fluid were much higher than Newtonian fluid.

Another striking fact from the comparison of two fluids were that away from contraction inlet, non-Newtonian fluid velocities near the inlet were almost zero; which was not the case for Newtonian fluid.

Polyacrylamide-water solution is chosen for this study because of the need to understand of the polymer melts processing such as injection molding which is very important for industry. Nowadays, powerful computers and better analyzing softwares can simulate the process with a minimum cost and minimum number of defective products which causes to win contracts in industry.

Finally the polyacrylamide is analyzed with FLUENT numerical simulation software for Reynolds numbers of 1, 10 and 100. All these simulations have shown that numerical simulation softwares are reliable and quickly simulates the problem with a minimum cost and without the need of any trial-error.

5.1 Recommendations for Future Studies

In this thesis, numerical simulations are compared with the experimental results of Fakilar, M.S. (2004) in which the working fluid is Newtonian fluid of water. Also the properties of the polyacrylamide solution in water are found from the literature review. It will be more realistic that comparing the numerical simulation of non-Newtonian fluid flow with the experimental study of non-Newtonian fluid flow. It is better to study both numerical and experimental at the same time with coordination with each other.

Also nowadays computers and numerical simulation softwares are capable of solving not only inelastic fluid flow but also the flow of viscoelastic fluids which will be more applicable and attractive for industry.

REFERENCES

- ALVES, M. A., OLIVEIRA, P. J., PINHO, F.T., 2003. Numerical simulation of viscoelastic contraction flows. Elsevier Science Ltd. Computational Fluid and Solid Mechanics, 1:2-4.
- ALVES, M. A., OLIVEIRA, P. J., PINHO, F.T., 2004. On the Effect of Contraction Ratio in Viscoelastic Flow Through Abrupt Contractions. J. Non Newtonian Fluid Mechanics 122.117-130.
- ANSYS- FLUENT Brochure
- ANSYS, Inc. – ANSYS 12.1 Release Brochure
- BROBOANA, D., MUNTEAN, T., BALAN, C., 2007. Experimental and numerical studies of weakly elastic viscous fluids in a hele-shaw geometry. The Publishing House of the Romanian Academy, 5-15.
- BOGER, D. V., 1982. Circular entry flows of inelastic and viscoelastic fluids. In advances in Transport Processes, ed. A. S .Mujumdar, R. A . Mashelkar, 2 : 43-98. New Delhi : Wiley Eastern
- BOGER, D.V., 1987. Viscoelastic flows through contractions. Ann. Rev. Fluid Mech. 19: J 57-82, 1:2:3:6:7-26.
- BOGUE, D. C., 1959. Entrance effects and prediction of turbulence in non-Newtonian flow. Ind. Eng. Chern. 5 1 (7) : 78-874.
- BOUSSINESQ, M.J., 1890 ,1981. Comptes Rendus, 9:49-113
- CARTER, T.R., (1969). Laminar Fluid Flow form a reservoir up to and through a tube entrance region, Ph.D. Thesis, University of Utah, Salt Lake City, Utah
- CHHABRA, R.P., RICHARDSON, J.F., 2008. Non-Newtonian Flow And Applied Rheology, 5-523
- CHU, C.S., (1965). MSc. Thesis, University of Tennessee.
- CHRISTIANSEN, E.B., KELSEY, S.J., CARTES, T.R., 1972. Laminar Tube Flow Through an Abrupt Contraction. Department of Chemical Engineering University of Utah, Salt Lake City, Utah. 372-379.
- COLLINS, M., SCHOWALTER, W. R. 1963. Behaviour of non-Newtonian fluids in the inlet region of a pipe. AIChE J. 9(1) : 9-804.
- COUTTE, M.M., 1890. Ann. Chem. Phys.(series 6), 21- 433.

- CROCHET, M. J., WALTERS, K., 1983. Numerical Methods in Non-Newtonian Fluid Mechanics. *Ann. Rev. Fluid Mech.* 15 : 60-241.
- CROCHET, M.J., DAVIES, A.R., WALTERS, K., 1984. Numerical Simulation of Non-Newtonian Flow, *Rheology Ser.*, Vol. 1 . Amsterdam : Elsevier. 352 pp.
- CUTTING EXTRUSION DIE DESIGN COSTS
- FAKILAR, M.S., 2004. Flow of a non-newtonian fluid through a contraction pipe. MSc Thesis, 11-46.
- FAN, C.T., HWANG, C.L., 1966. Bibliography of hydrodynamic entrance region flow, *Kansas State University Bull.*, 3-50.
- FESTER, V., MBIYA, B., SLATTER, P., 2008. Energy Loses of Non-Newtonian Fluids in Sudden Pipe Contractions. *Chemical Engineering Journal* 145, 57-63.
- FREIEDMANN, M., GILLIS, J., LIRON, N., (1968) *Appl. Sci. Res. A* 19-426.
- GRILLET, A.M., YANGB, B., KHOMAMI, B., SHAQFEH, E.S.G., 1999. Modeling of viscoelastic lid driven cavity flow using finite element simulations. *J. Non-Newtonian Fluid Mech.* 88, 99-131.
- HAGENACK, E., 1860. *Pogg, ANN.*, 109-385.
- HOLLAND, F.A., BRAGG, R., 2002. *Fluid Flow For Chemical Engineers.* Butterworth-Heinemann Linacre House, Jordan Hill, Oxford, 49-375.
- ISAKSSON, P., RIGHDAL, M., 1994. Numerical simulation of blade coating with short dwell and roll application coater. *Rheol. Acta*, 33-454.
- JAMES, D.F., SARINGER, J.H., 1980. Extensional Flow Dilute Polymer Solutions. *Journal of Fluid Mechanics*, 655-671.
- KESTIN, J., SOKOLOV, M., Wakeman, W., 1973. *Appl. Sci. Res.*, 27-241
- KEUNINGS, R., 2001. Advances in the computer modeling of the flow of polymeric liquids. Published in *Comp. Fluid Dyn. J.*, Vol. 9, 449-458, 1:2-20.
- KIM, J.H., OZTEKIN, A., NETI, S., 2000. Non-linear dynamics of viscoelastic flows in axisymmetric conduits with abrupt changes. Department of Mechanical Engineering and Mechanics, Lehigh University, Bethlehem, PA 18015, USA, 1:2-25.

- KIM-E, M.E., BROWN, R.A., ARMSTRONG, R.C., 1983. The Roles of Inertia and Shear Thinning in flow of an Inelastic Liquid Through an Axisymmetric Sudden Contraction. Elsevier B.V.
- KUNDU, P.K., COHEN, I.M., 2008. Fluid Mechanics, Elsevier, The Boulevard, Langford Lane Kidlington, Oxford, OX5 1GB, UK, 418-901.
- LEAL, L.G., OBERHAUSER, J.P., 2000. Non-Newtonian fluid mechanics for polymeric liquids: A status report. Departments of Chemical** and Materials Engineering University of California, Santa Barbara Santa Barbara, California 93106, USA, 1-6.
- LIU, G.R., 2002. Mesh Free Methods, CRS Press,.
- MENDELSON, M.A., YEH, P.W., BROWN, R.A., ARMSTRONG, R.C., 1982. Approximation error in finite element calculation of viscoelastic fluid flow. J. Non-Newtonian Fluid Mech., 10-31.
- MITSOULIS, E., ZISIS, Th., 2001. Flow of Bingham plastics in a lid-driven square cavity. J. Non-Newtonian Fluid Mech., 101:173–180.
- MURTHY, R.A.V., BOGER, D.V., 1971. Developing Velocity Profiles on the Downstream Side of a Contraction For Inelastic Polymer Solutions. Department of Chemical Engineering, Monash University, Clyton, Victoria, Australia. 1.
- OLIVEIRA, M.S.N., ALVES, M. A., OLIVEIRA, P. J., PINHO, F.T., 2007. Effect of Contraction Ratio Upon Viscoelastic Flow in Contractions: The Axisymmetric Case.. J. Non Newtonian Fluid Mechanics 147.92-108.
- REDDY, M.P., REDDY, J.N., 1992. Finite element analysis of flows of non-Newtonian fluids in three-dimensional enclosures. Int. J. Nonlinear Mech. 27.
- SCHMIDT, F. W., ZELDEN, B., 1969. Laminar flows in inlet sections of tubes and ducts. AIChE J. 15 (4) : 14-612.
- SHAMEKHI, A., SADEGHY, K., 2009. Cavity flow simulation of Carreau–Yasuda Non Newtonian fluids using PIM meshfree method. Applied Mathematical Modelling 33.4131–4145, 1-15.

- SONG, J.H., YOO, J.Y., 1987. Numerical Simulation of Viscoelastic Flow Through a Sudden Contraction Using a Type Dependent Difference Method. Elsevier B.V.
- SOUSAA, P.C., COELHOB, P.M., OLIVEIRAA, M.S.N., ALVES, M. A., 2009. Three-dimensional flow of Newtonian and Boger fluids in square-square contractions. *J. Non-Newtonian Fluid Mech.* 160.122–139, 1:2-124
- STOLARSKI, T., NAKASONE, Y., YOSHIMOTO, S., 2006. Engineering analysis with ansys software. Elsevier Butterworth-Heinemann. 7-473.
- SUNTAR, P., Polymer Rheology. Department of Chemical Engineering, Indian Institute of Technology (IIT) Bombay, Mumbai, 2:3-21.
- TANGUY, P.A., FORTIN, A. F., BERNARD, A., 2001. 3-D finite element model for polymer melt flow. *Adv. Poly. Technol.* 8, 99–113.
- VRENTAS, J. S., DUDA, J. L., BARGERON, K.G., 1966. Effect of axial diffusion of vorticity on flow development in circular conducts. *AIChE J.* 12(5) : 44-837.
- VRENTAS, J.S., DUDA, J.L., 1973. Flow of a Newtonian Fluid Through a Sudden Contraction. The Dow Chemical Company Midland, Michigan, U.S.A. 241-260.
- WEISSBERG, H.L., 1962. End correction for slow viscous flow through long tubes. *Phys. Fluids* 5(9) : 36-1033.
- WUNDERLICH, A.M., BRUNN, P.O., DURST, F., 1988. Flow of Dilute Polyacrylamide Solutions Through a Sudden Planar Contraction. Elsevier B.V.
- YAMAMOTO, T., ISHIYAMA, M., NAKAJIMA, M., NAKAMURA, K., MORI, N., 2003. Three-dimensional viscoelastic flows through a rectangular channel with a cavity. *J. Non-Newtonian Fluid Mech.* 114. 13–31, 1-19.
- YANG, M.H., 2001. Flow Behavior of Polyacrylamide Solution. III. Mathematical Treatment. Department of Chemical Engineering, Kao Yuan Institute of Technology, Kaohsiung County, 82101 Taiwan, Republic of China. 1-6.
- YAVUZ, M.M., 2011. Numerical analysis of vortex breakdown of unmanned combat air vehicles, MSc Thesis, 18-57.

- YESILATA, B., OZTEKIN, A., NETI, S., 1999. Instabilities in Viscoelastic Flow Through an Axisymmetric Sudden Contraction. J. Non Newtonian Fluid Mechanics 85.35:62.
- YESILATA, B., OZTEKIN, A., NETI, S., 2000. Non-isothermal Viscoelastic Flow Through an Axisymmetric Sudden Contraction. J. Non Newtonian Fluid Mechanics 89., 133-164.

CURRICULUM VITAE

Münire ÖZTÜRK was born on January 22, 1987 in Adana, TURKEY. She graduated from Adana Seyhan Rotary Anatolian High School in 2005 and started under graduate program of Mechanical Engineering Department of Çukurova University in same year. She graduated from Mechanical Engineering Department in 2009 then she began graduate program of energy section of Mechanical Engineering Department at Çukurova University. She is married and has a son.