TWO DIMENSIONAL FLOW OF SHEAR THINNING FLUID AROUND A SQUARE CYLINDER

A THESIS SUBMITTED IN THE PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

Bachelor of Technology

In

Chemical Engineering

Under the supervision of

Prof. Akhilesh Kumar Sahu

By **Bishmaya Kumar Rout**

(111CH0623)

DEPARTMENT OF CHEMICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA-769008, ODISHA

CERTIFICATE

This is to certify that the thesis entitled, "**TWO DIMENSIONAL FLOW OF SHEAR THINNING FLUID AROUND A SQUARE CYLINDER"**, presented by **Bishmaya Kumar Rout** in partial fulfillments for the requirements of the award of Bachelor of Technology Degree in Chemical Engineering at National Institute of Technology, Rourkela, has been carried out under my supervision and guidance.

May 7, 2015

Prof. Akhilesh Kumar Sahu

Department of Chemical Engineering

National Institute of Technology

Rourkela-769008,Odisha

ACKNOWLEDGEMENTS

I would like to express my sincere gratitude and genuine appreciation to Prof. Akhilesh Kumar Sahu, my project guide, for his invaluable help, continuous and indomitable encouragement throughout my project work. This project wouldn't have been possible without his valuable feedback, kind encouragement, support and timely suggestions during this entire duration of this work.

I would thank my HOD, professors, seniors and dear companions for helping me in this thesis work. I also thank my lab mates Amogh, Pritish and Eugine for helping me out in the lab. I would like to thank Miss Trupti Patil for all the software help. I would also like to thank Mr.Trupti Nayak and Payal for always being there for me.

Last but not the least, the best wishes, love and blessings of my parents and other family members had been a constant source of inspiration and encouragement throughout my project work.

May 7, 2015 ------------------------

NIT Rourkela

Table of Contents

Abstract

The current thesis has been segregated into four parts. The first part is to observe the lid driven flow in a square cavity using water as the material. The simulation was carried out at different Reynolds number (100<Re<500) and the velocity streamlines and vectors plots were plotted. The second part is to study the fluid flow across a square cylinder for a Non-Newtonian fluid in the range of 20<Re<100 and plot the x velocity along the vertical centre line and y velocity along the horizontal centre line. For shear thinning of the fluid the power law index was taken to be n<1. The results were matched with that of literature. The third part is the study of momentum transfer from a square cylinder and determination of drag coefficients. The drag values for Newtonian and shear thinning fluids were calculated and the results were validated with the literature. The Reynolds number range was from 1<Re<100 and n<1. The fourth part is to study the two dimensional unsteady laminar flow of a Newtonian and power law fluid across a square cylinder. The Strouhal number, Average Drag Coefficient and RMS value of lift were calculated and then compared with the literature. The effect of Reynolds number on Strouhal number, average drag and root mean square lift coefficient was carried out. The Re range was from 60<Re<160, beyond which the flow goes three dimensional which isn't covered in this project. The vortex shedding at different intervals of time is also shown.

Keywords: Strouhal number, Drag Coefficient, Lift Coefficient, Vortex Shedding, Power-law fluids.

List of Figures

List of Tables

Nomenclature

μ	Viscosity of the fluid in (Pa s)
u	x-component of velocity (m/s)
v	y- component of velocity (m/s)
ρ	Density of the fluid (kg/m^3)
F	Body forces acting on the fluid (N)
A	Area of cross-section (m^2)
F_D	Drag force acting on the body (N)
F_L	Lift force acting perpendicular to the cylinder motion (N)
f	Vortex Shedding frequency (1/s)
P	Pressure (Pa)
C_D	Drag Coefficient
C_{Lrms}	Root mean square value of lift coefficient
St	Strouhal number
n	Power law index
k	Consistency index (Pa s^n)
Re	Reynolds number
τ	Shear stress
U_∞	Inlet velocity (m/s)

Chapter 1

Introduction

Bluff bodies are the structures with shapes that fundamentally disturb the flow around them, rather than stream around a streamlined body. The vital illustrations in this category incorporate circular cylinder, square cylinder and rectangular cylinder. The flow around circular and square cylinders is usually concentrated on in light of their wide applications. For the past hundred years, the flow around circular and hollow bluff bodies has been the subject of exceptional research principally attributable to the building essentialness of structural design, stream affected vibration, and acoustic outflows. Lately, such studies have gotten a lot of consideration because of change in experimental measurement strategies.

Structures that have rectangular or close rectangular cross segments incorporate engineering highlights on structures, beams, fences and backings inside and outside stream geometry. At the cross stream point these structures result in partition from the upper and lower piece of the body. As a result of unreliability the vortex shedding is known as von Karman Vortex Street. Right when scaled with the cross stream bluff body measurement and approaching velocity magnitude, the critical Reynolds number where the vortex shedding begins is of the order of 50 for a zero angle of incidence.

In the present work investigation was constrained to Reynolds number under 150, underneath Re under 160 the stream stays laminar in nature. The vortex shedding is described by an extremely defined frequency. The frequency and the wake behavior rely on diverse parts of stream field, for example, the top and bottom conditions, blockage proportion of the stream section, upstream speed and the aspect ratio of the structures.

1.1 Newtonian Fluid: A fluid for which on plotting stress versus strain rate curve, passes through the origin and is linear in nature is known as a Newtonian fluid. Mathematically, it can be defined as,

$$\tau_{ij} = \mu\left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i}\right) \quad (1.1)$$

Where,

τ is the shear stress given by the fluid.

μ is the fluid viscosity.

1.2 Non-Newtonian fluid: It is a fluid whose stream properties are different those of Newtonian fluids. Most commonly the viscosity of Non -Newtonian fluids is not independent of shear rate. In case of these liquids, the plot drawn between the shear rate and stress is different and can be time independent.

1.3 Computational Fluid Dynamics: Fluid mechanics is the branch of physics which involves the study of fluids and the forces on them. CFD is a section of fluid mechanics which makes use of numerical methods and complex algorithms to solve fluid flow problems. The advantages of employing CFD are:

- It predicts performance before modifying or installing systems.
- It saves cost and time.
- It is dependable and trustworthy tool for design and analysis.
- It provides information about HVAC parameters.
- Other applications include automobile design, weather science, civil engineering, etc.

1.4 ANSYS: It is an engineering simulation software. It is the solver of CFD of choice for complex flows ranging from incompressible to mildly compressible to highly compressible flows. The current project uses ANSYS 15 to simulate the fluid flows.

1.5 Drag Coefficient: In the context of fluid dynamics, the drag coefficient (abbreviated as C_d) is defined as a quantity which is dimensionless and is used to quantify the object's drag or resistance in a liquid environment. The drag characteristic of an object in case of fluid flow is very essential to understand, especially for engineering aspects. Such information can be utilized to decrease the drag on autos, air ships and structures.

1.6 Lift Coefficient : The lift coefficient abbreviated as C_L is dimensionless and relates the lift generated by a lifting body to the density of the fluid along the body, velocity of the fluid, and the concerned area of cross-section. It is given by:

$$C_L = \frac{2F_L}{\rho U^2 A}$$

1.7 Strouhal Number: Strouhal number is found out from the expression:

$$St = \frac{fD}{U}$$

Where, St represents strouhal number,

f= vortex shedding frequency (1/s)

D= Cavity diameter (m) and U= inlet velocity of the fluid (m/s).

The frequency,f is taken to be the frequency of one lift cycle and multiplying with the time step.

Chapter 2

Literature Review

This section briefly lists the previous studies available in the literature on the two dimensional shear thinning flow over a square cylinder. Sharma and Eswaran [6] employed the finite volume method and used SIMPLEC code and non- staggered grid to solve laminar and turbulent fluid flow over a cylinder. The stream transfer phenomena in the forced and mixed regimes for a cylinder in Newtonian fluids have been numerically studied by Breuer et al. [7] and Turki et al. [8]. Sohankar et al. [9] found that vortex shedding begins at Re>50, Okajima et al. [10] determined the wake frequency and Strouhal number for different value of Reynolds number for square cylinders, Davis and Moore [11] investigated the phenomena of vortex shedding, lift,drag and Strouhal number for higher value of Reynolds number. , Dhiman et al. [4] studied the variation of Prandtl number and effect of blockage ratio on the rate of heat transfer ,etc. Chhabra et al.[3] studied behaviour of power law fluids on momentum and heat transfer characteristics of an inclined square cyinder in steady regime. Sharma et al [14] presented a review on the CFD analysis of two dimensional steady flow around a square cylinder. The wake characteristics for laminar flow were observed. The effect of time steps, upstream and downstream degree, blockage and dispersion of lattice points were thoroughly investigated by Sohankar et al. [9]. Saito et al.[16] compared the various numerical algorithms for a flow around a square cylinder.

Sahu et al. [15] investigated two dimensional unsteady laminar flow around a square cylinder. They were numerically investigated in the range of $60<Re<160$ and $0.5<n<2$. The drag coefficients, Strouhal number and the rms value of the lift has been obtained for the above range of conditions. The periodic value was taken into consideration and the Reynolds number was varied according to the power law index.

2.1 Objectives of the Present Work

In the present work, two dimensional fluid flow across a square cylinder has been studied. The total project is divided into four parts. The first part is to observe the lid driven flow in a square cavity using water as the material. The simulation was carried out at different intervals and the velocity streamlines and vectors plots were plotted. The second part is to study the fluid flow across a cavity for a Non-Newtonian fluids in the range of $20<Re<100$ and plot the x velocity along the vertical centre line and y velocity along the horizontal centre line. The results were compared with the literature. The third part is the study of momentum transfer from a square cylinder and determination of drag coefficients. The square cavity was taken for this problem and drag values for Newtonian and shear thinning fluids were calculated and the results were validated with the literature. The fourth part is to study the 2D unsteady flow of a Newtonian and power law fluid along a square cylinder. The Strouhal number, Average Drag Coefficient and RMS value of lift were calculated and then compared with the literature.

Chapter 3

Mathematical Formulation

The physical problem considered here is two dimensional flow of a fluid around a square cylinder of size B(1m), placed in a uniform stream of speed. The representing mathematical equations for a liquid flow around a square cylinder are the continuity equation, and momentum equations. The flow along a square/circular cylinder has been simulated by solving for an incompressible fluid in 2D geometry. The governing equations are as follows:

Continuity Equation: $$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad (3.1)$$

X-Momentum Equation:

$$\frac{\partial u}{\partial t}+\frac{\partial (uu)}{\partial x}+\frac{\partial (uv)}{\partial y}=-\frac{\partial P}{\partial x}+\frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \qquad (3.2)$$

Y-Momentum Equation:

$$\frac{\partial v}{\partial t}+\frac{\partial (vv)}{\partial y}+\frac{\partial (uv)}{\partial x}=-\frac{\partial P}{\partial y}+\frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \qquad (3.3)$$

The boundary conditions applied to the geometry are:

1) Symmetry at y=0, y= H.

$$\frac{\partial u}{\partial y}=0 \ ; \quad v=0$$

2) No slip condition is applied to the surface of cylinder.

$$U_o=0; \ v_o=0$$

3) The inlet boundary is given by a uniform x velocity with zero flow in the lateral direction. i.e.,

$$U_\infty=0.5;\ v_o=0$$

4) At the outlet boundary the default case in FLUENT is provided, which assumes a zero diffusion flux for all the flow variables.

$$\frac{\partial \emptyset}{\partial x} = 0$$

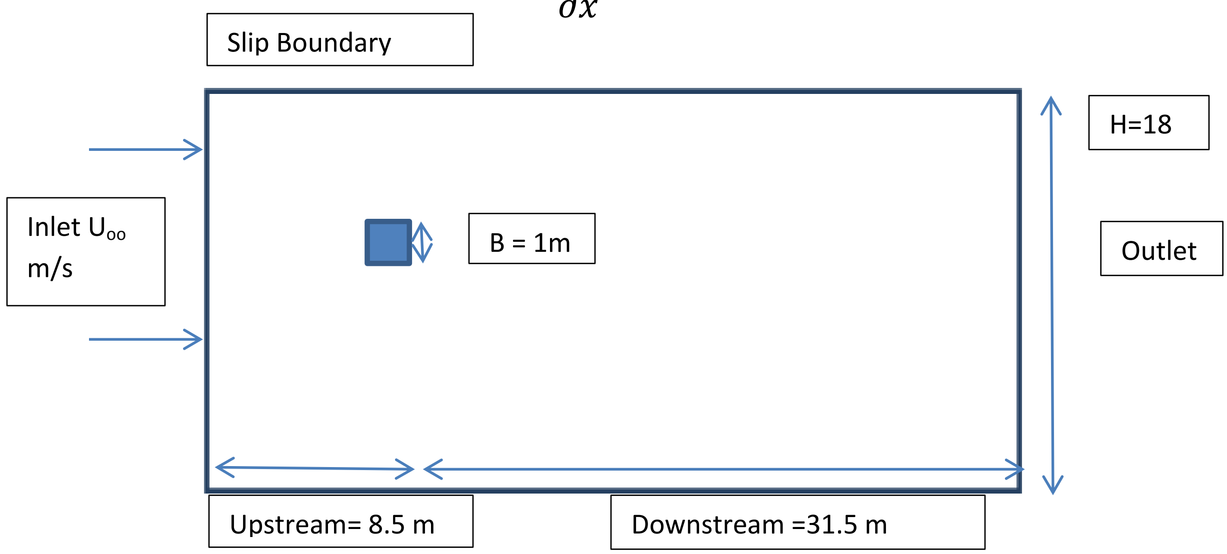

Fig 3.1 Schematic of a flow along a square cylinder

Reynolds Number:

It is defined as a dimensionless number that gives inertial forces to viscous forces ratio and furthermore evaluates the relative significance of these two type of forces for given stream conditions. It is given by:

$$Re= \frac{\rho VD}{\mu} \quad (3.4)$$

Where,

Re= Reynolds number

ρ= density of the fluid,

D= Characteristic Length.

V= Characteristic Velocity of the fluid.

μ= Viscosity of the fluid.

Drag Coefficient:

The drag coefficient C_D is defined as:

$$C_D = \frac{2F_D}{\rho U_\infty^2 A} \tag{3.5}$$

Where,

F_D = drag force,

ρ= mass density of the fluid,

A= area of cross section

$U_\infty{}^2$= Relative speed of an object w.r.t fluid

It is not a fixed constant for any given body shape. The value changes with the airflow speed and with the change in the value of Re.

Strouhal number is found out from the expression:

$$St = \frac{fD}{U_\infty} \tag{3.6}$$

Where, St represents strouhal number,

f= vortex shedding frequency (1/s)

D= Cavity diameter (m) and U_∞= inlet velocity of the fluid (m/s).

The frequency, f is taken to be the frequency of one lift cycle and multiplying with the time step.

The average drag coefficient, C_{Davg} is found out by taking any two cycles and calculating the average value of the drag in those cycles. Similarly the RMS value of lift,C_{Lrms} is calculated by taking a single cycle and finding out the square root of the average square of the lift values in that cycle.

3.1 Numerical Methodology

The computational domain is shown in fig 3.1. The square cylinder of length 1 m is placed in a rectangular domain of length 40 m and height 18 m. The velocity of the fluid at the inlet is fixed to the value of 1/0.5 m/s. The mesh generated using ANSYS 15. The node distributions in all the edges were specified in order to give a better control on the grid distribution. The number of nodes distributed on each surface of the cylinder was 100 and that on the top and bottom walls were 400 and on the inlet and outlet were 200. Inflation was done to get a finer mesh. The mesh generated is shown in Fig 3.2.

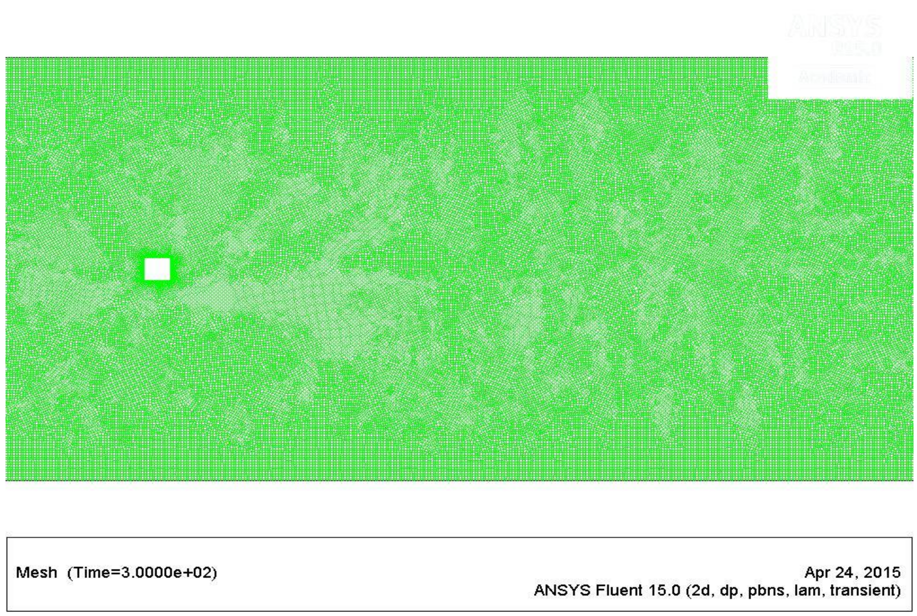

Fig 3.2 Mesh generated for the square cylinder

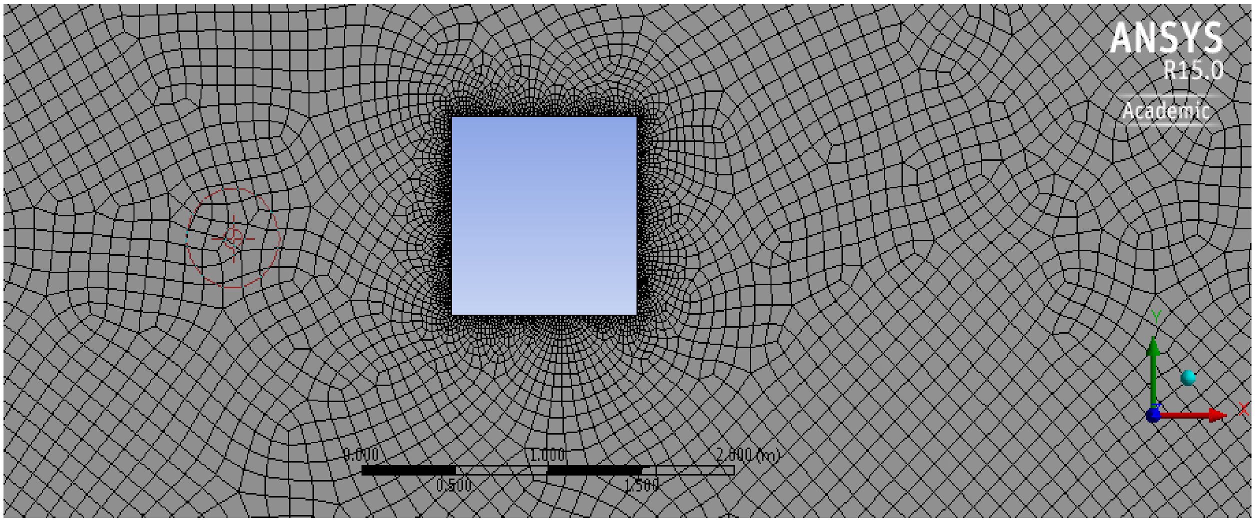

Fig 3.3 Closer view of Mesh around the square cylinder

Fluent: The analysis was done in fluent by importing the meshed file that is generated in ANSYS. The steps which was followed are given below which includes all the conditions and the boundaries values for the problem statement.

1) *Checking of mesh and scaling:* The fluent solver 2D is selected and the meshed file is imported. Scale is scaled to meters.
2) *Solver and material selection:* The solver is characterized first. Solver is based on pressure and plan as implicit, and time as unsteady/steady depending on the problem statement.
3) *Material Selection and Operating Condition:* Density was fixed at 1kg/m^3. For Newtonian fluids the viscosity was changed according to the Reynolds number. For Non-Newtonian liquids, non- Newtonian power law was selected for viscosity.

Consistency index, k was calculated from the Reynolds number expression for unsteady state.

$$Re= \frac{\rho V^{2-n} D^{n}}{k} \qquad (3.6)$$

Where,

n= power law index, D= Characteristic Length

4) *Boundary Conditions:* Velocity Inlet was specified for the inlet boundary and the value of velocity magnitude in the streamline direction was specified.

Top and Bottom walls: Symmetry is specified for the sides, top and bottom walls. No slip condition was applied to the walls. But for unsteady flow, “Wall” was selected and no shear condition was selected to ensure that the shear components were zero.

Outlet: The outlet boundary was set as “pressure outlet” for unsteady and “outflow” for steady flow over a square cylinder.

The reference values were computed from the inlet. The standard discretization method was used for the pressure interpolation scheme and QUICK scheme discretization method was used for momentum. Results are obtained iteratively utilizing the segregated solver with SIMPLE algorithms to do this. For unsteady flow, second order transient method was selected.

5) Residual monitor and convergence criteria were set at 1e-6. Drag and lift monitors were also set. All the values were saved by selecting the “Write” option. For unsteady flow, the flow symmtery was broken and the phenomena of vortex shedding was eventually triggered by artificial perturbation. The flow along then cavity was disturbed by patching a velocity in x direction of 1 m/s and 0 m/s in the upper and bottom part of domain. This is done using a custom field function defined as,

$$u=\frac{y+|y|}{2y} \tag{3.7}$$

where,

u is the velocity in x direction.

This function is saved in the case file. After initializing the flow, the x velocity of the entire fluid zone is patched with the function. The use of this function is not mandatory. It was applied here to break the symmetry of the geometry and start the onset of vortex shedding. For steady flow the number of iterations was specified and no unsteady a time step for 0.3 was used for all the calculations. The maximum time step and the number of iterations were fixed at 100 and 1000 respectively.

Chapter 4

4. Results and Discussions

4.1 Lid Driven Flow in a Square Cavity

To observe the fluid flow in a cavity using FLUENT (using different Reynolds number). The cavity has three stationary walls and one moving wall. Material to be used is water-liquid. Carry out proper iterations at different Reynolds number using the same material. Plot and display the velocity streamlines and vectors.

The present study aims at studying the fluid flow in the square cavity for a set of Re (100,200,500). Water specifications are:

Density of the fluid = 998.2 kg/m^3, Viscosity = 0.001003 kg/m-s,

Thermal Conductivity =0.6 W/mK, Specific Heat Capacity= 4182 J/kgK

The solutions were converged for different number of iterations depending upon the Reynolds number. The plots for stream function and vector are shown below.

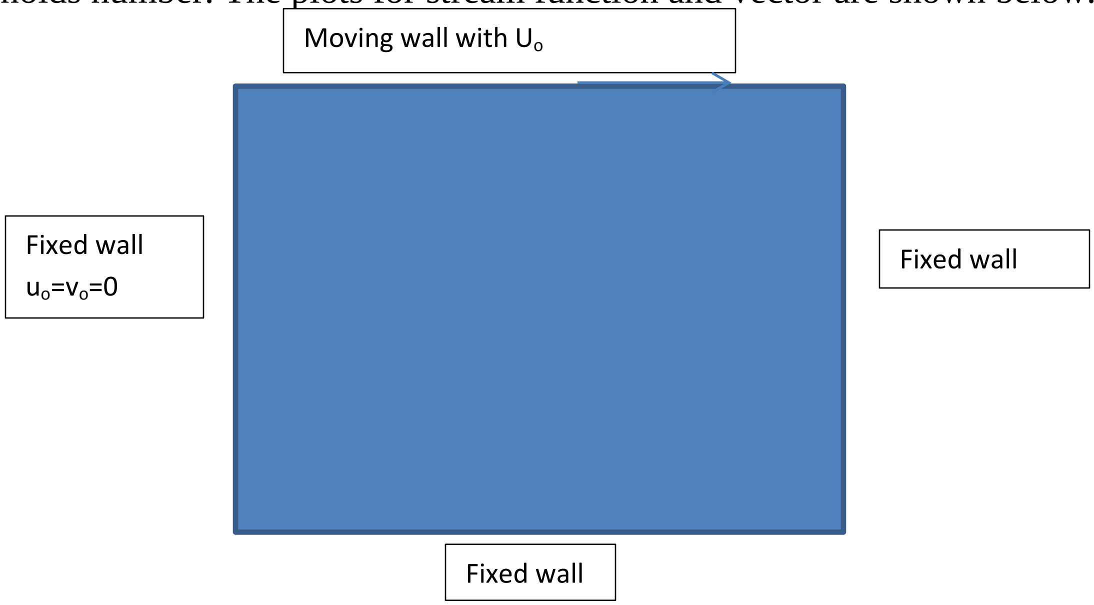

Fig 4.1 Square Cavity

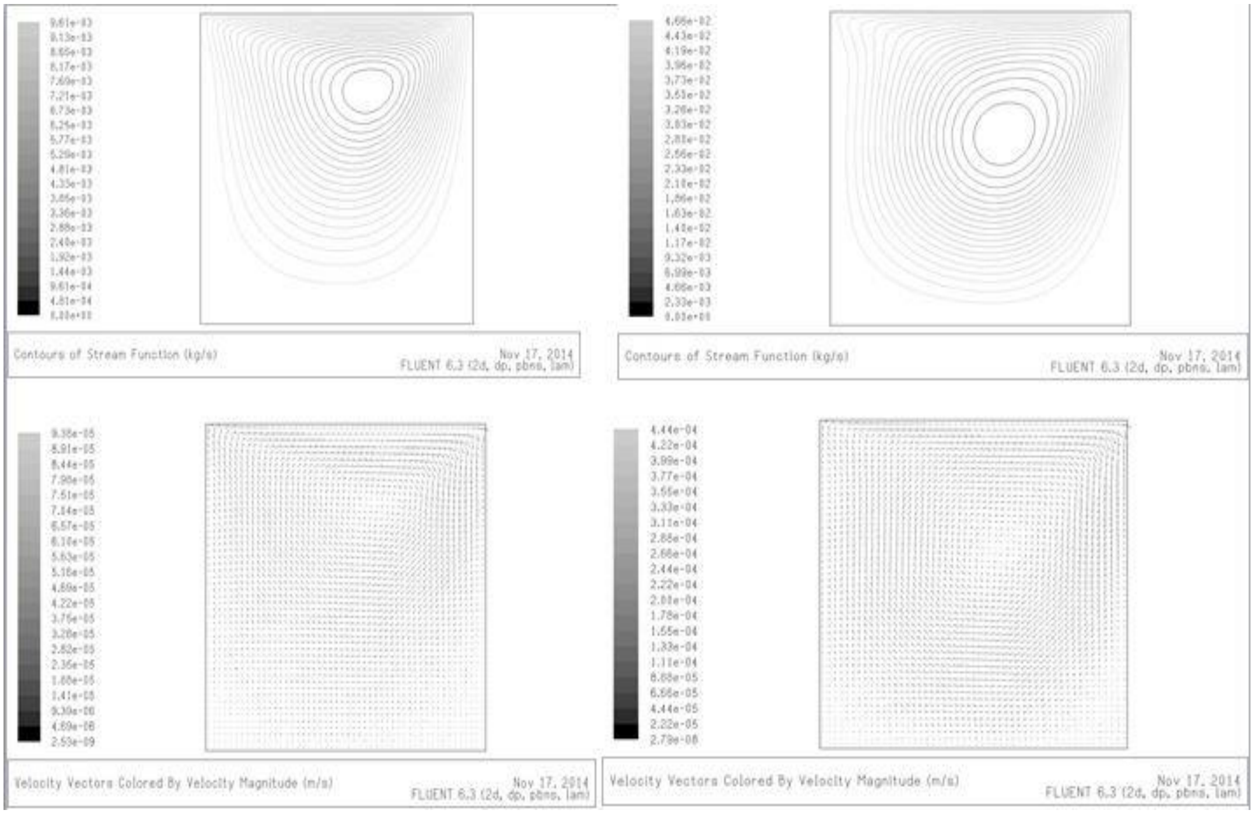

Fig 4.2 (a) Instantaneous Streamlines for Re 100 and Re 500 (b) Vector Plots for Re 100 and 500

Increasing the Reynolds number accounts for more turbulence in the flow patterns. It proves that the threshold of turbulence is dependent on the nature of media. On increasing the Reynolds number the velocity profile seems to be unclear at the edges due to absence of grid points. It can be resolved by running the simulation with a finer mesh.

4.2 Steady State Flow Around a Square Cylinder

Study the flow of fluid along a square cylinder in two dimensions for a Non-Newtonian shear thinning fluid with power law index, n<1, and for Re in the range of 20-100 for steady state conditions. Plot the velocities in x and y direction along the central axis in the vertical and horizontal direction respectively. Compare data with the literature. (Ghia et al.[1], Neoftou[2])

Panagiotis [2] exhibited a numerical technique where the QUICK and SIMPLE methods are applied for explaining Non-Newtonain viscous streams. For approving the technique, the lid driven hole stream problem is used and the outcomes contrasted with the results available in literature. Ghia et al. [1] studied the flow in a square cavity using an implicit multi-grid method by using the vorticity stream formulation of two dimensional Navier Stokes equation.

A lid driven cavity as shown in fig 3.1 was taken for simulation. The geometry and mesh was defined. For non- Newtonian viscous flow the Reynolds number was calculated with n=0.5. A new line was created and along the horizontal and vertical centre line and the x and y velocities were found out. The results on comparing with that of Neoftou [2] yielded the same results.

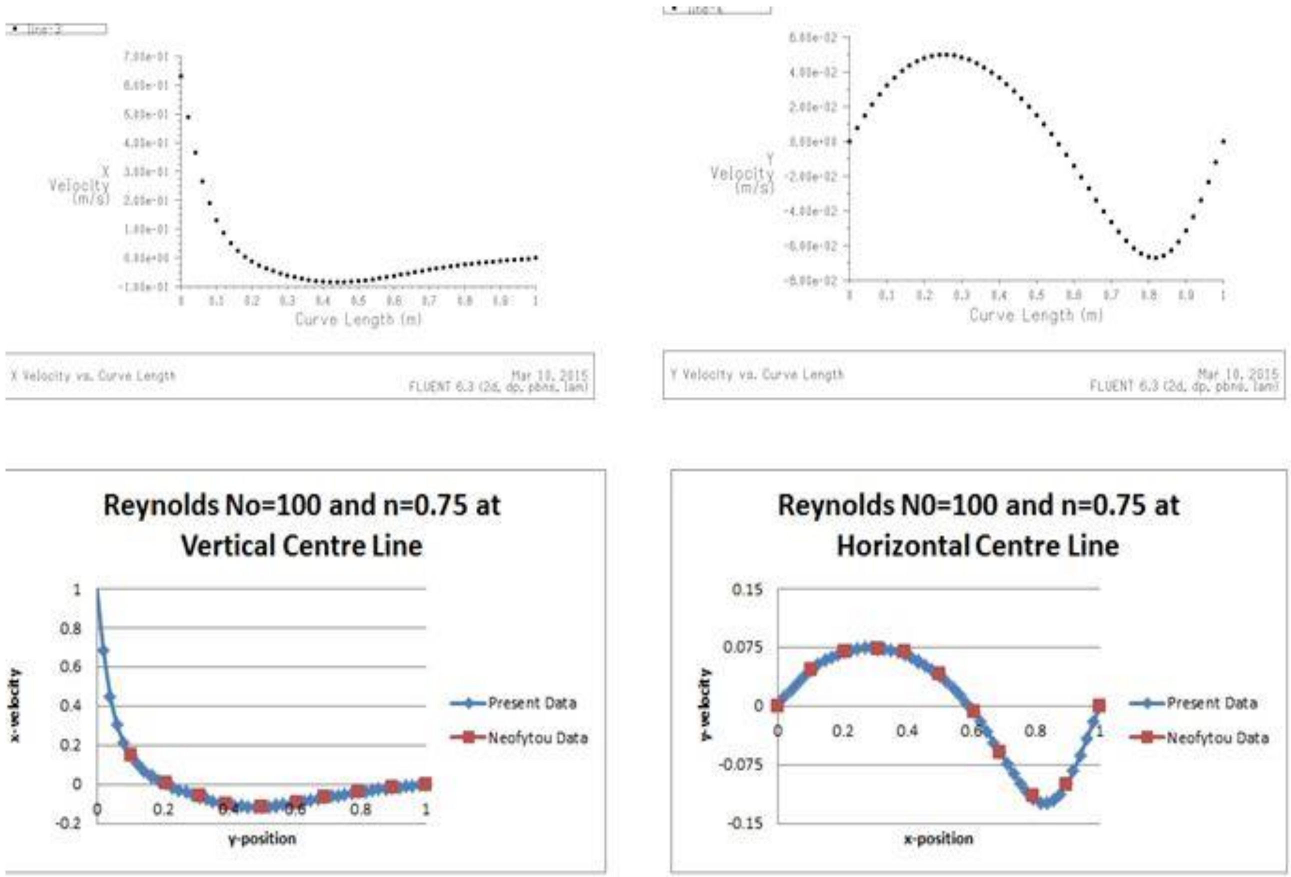

Fig 4.3 (a) Plot of x velocity along vertical center axis (b) y velocity along horizontal axis (c)Comparison of Current data along vertical center axis (d) along horizontal center axis

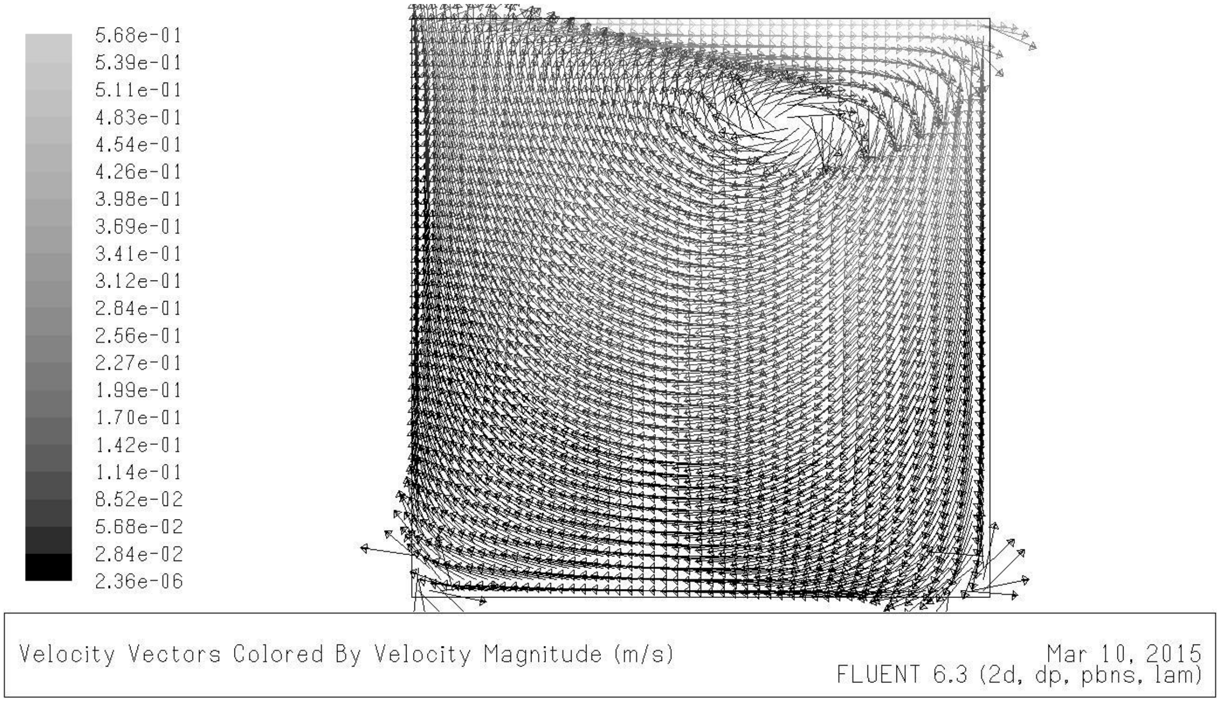

Fig 4.4 Velocity vectors for the square cavity

The vector plots clearly demonstrated the nature of fluid flow inside the cavity by showing the start of wake formation in the corners of the cavity at the bottom. It was observed that both the data are in well agreement to each other. The obtained center line velocity was compared with the ones in Neofytou [2] and it was encouraging to see the values in perfect alignment with the ones in the literature.

4.3 Steady Flow Over a Square Cylinder for Newtonian and Non-Newtonian Fluids

Study of steady flow over a 2D square cylinder (Fig 4.1). Problem domain is a rectangular chamber with a square cylinder in it. The diameter being 1 m and the inlet velocity be 1 m/s for steady flow and 0.5 m/s for unsteady flow. Exit pressure is taken as 0 Pa. Simulate the flow for Re 1,5 and 40 for Newtonian fluids and Re 10,20 and 100 for Non Newtonian shear thinning fluids. We are interested in finding out the drag coefficients and comparing it with the values present in literature.

The square cavity with the dimensions shown in fig 4.1 was taken. The governing equations are the continuity and the momentum equations. The boundary conditions mentioned in chapter 4 were the same. Various studies have been done in literature on comparing drag and lift coefficients for viscous flows. Chabbra et al [3] studied the characteristics of square cylinder immersed in liquids with respect to heat transfer and momentum. Dhirman et al [4] , Paliwal et al [5] carried out the computations for Newtonian and non - Newtonian viscous flows. Sharma and Eswaran [6] employed the finite volume method and used SIMPLEC code and non- staggered grid to solve laminar and turbulent fluid flow over a cylinder.

The drag coefficients have been formulated in chapter 3 for both Newtonian and Non-Newtonian fluids. Validation of results:

Table 4.1 Comparison of C_D in a steady flow (Newtonian fluid)

Source	Re=1	Re=5	Re=40
Present	14.32	4.81	1.69
Ref [4]	11.71	4.46	1.67
Ref [3]	14.34	4.83	1.77
Ref [6]	14.34	4.85	1.76
Ref [5]	-	4.81	1.90

Table 4.2 Comparison of C_D in a steady flow (Non-Newtonian fluid)

Source, index n =0.5	Re=10	Re=20	Re=100
Present	3.30	2.32	1.47
Ref [3]	3.34	2.15	-
Ref [5]	3.43	2.33	-
Ref [4]	4.09	2.26	-

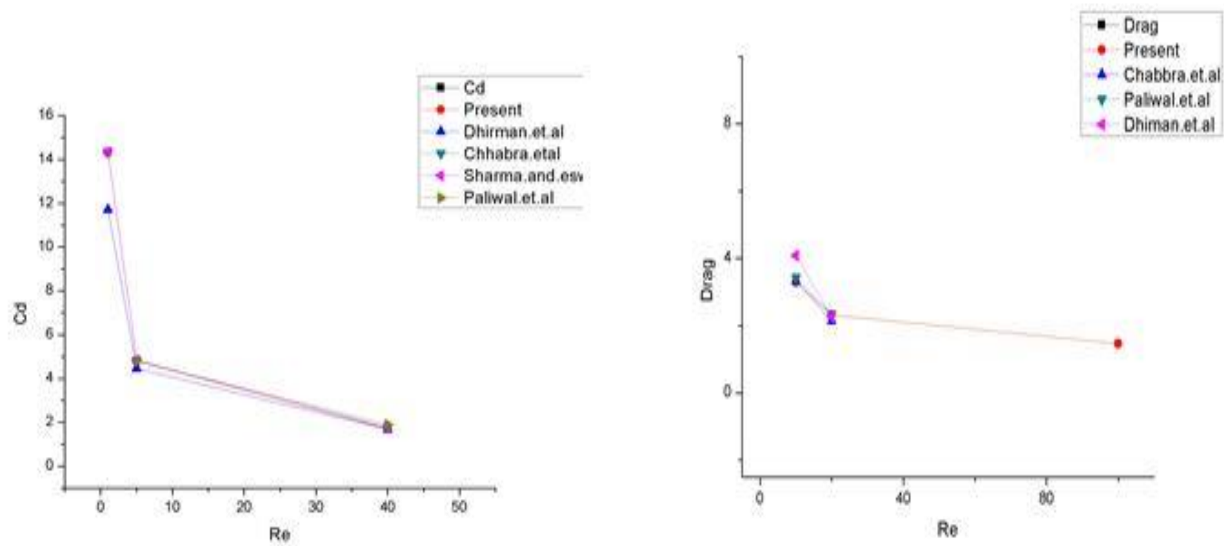

Fig 4.5 Comparison of Drag Values for (a) Newtonian (b) Non-Newtonian fluids

On comparing the drag coefficients of present data with the literature, it was found that the results were in agreement.

4.4 Unsteady Flow Over a Square Cylinder for Newtonian and Non-Newtonian fluids)

Study of the unsteady flow over a 2D cylinder. Problem domain is a rectangular chamber with a square cylinder of 1m dimension in it. Fluid properties are density= 1 kg/m^3, inlet velocity of 1m/s for unsteady Newtonian flow and 0.5 m/s for unsteady non-newtonian flow. Exit pressure is taken as 0 Pa. We are interested in capturing vortex shedding behind the cylinder and calculate its frequency or Strouhal number. We would also want to find the lift and drag coefficient, average drag and Root mean square lift coefficient value.

The computational domain for the above problem statement is mentioned in chapter four. (fig 4.1). The governing equations and boundary conditions are the same as mentioned earlier. The outlet boundary condition for unsteady flow is “pressure outflow”. The symmetry of the stream and inevitably initiate the vortex shedding, artificial perturbation was applied. The flow was disturbed by patching a uniform x velocity of 1m/s and 0m/s in the upper and lower section of the domain respectively. This is done using a custom field function defined as,

$$u= \frac{y+|y|}{2y}$$

where, u is the x direction velocity.

The uniform field function is saved in the case file with the name initial velocity. After the flow is initialized, the velocity in the x direction is patched with this custom field function for the entire zone of fluid.

Strouhal number is found out from the expression:

$$St=\frac{fD}{U}$$

Where, St represents strouhal number,

f= vortex shedding frequency (1/s)

D= Cavity diameter (m) and U= inlet velocity of the fluid (m/s).

The frequency,f is taken to be the frequency of one lift cycle and multiplying with the time step.

The average drag coefficient, C_{Davg} is found out by taking any two cycles and calculating the average value of the drag in those cycles. Similarly the RMS wale of lift,C_{Lrms} is calculated by taking a single cycle and finding out the square root of the average square of the lift values in that cycle.

Table 4.3 Effect of Re on unsteady flow around a square cylinder (Newtonian)

Re	St	C_{Davg}	C_{Lrms}	C_D
60	0.112	1.570	0.070	1.585
75	0.128	1.524	0.105	1.528
100	0.134	1.464	0.157	1.468
120	0.136	1.415	0.208	1.423
150	0.145	1.411	0.252	1.420

Table 4.4 Effect of Re on unsteady flow around a square cylinder(Non-Newtonian)

Re	St	C_{Davg}	C_{Lrms}	C_D
60	0.14814	1.458569	0.011585	1.4639
75	0.15503	1.375782	0.028625	1.37956
100	0.15872	1.470917	0.274555	1.47879
120	0.13605	1.740949	0.822868	1.58207
150	0.12345	1.824452	0.913962	1.87898

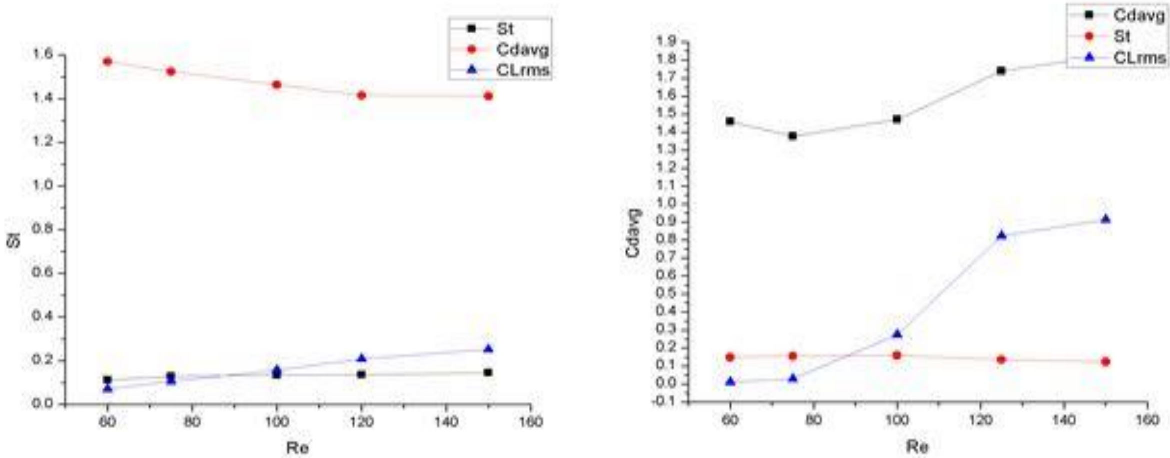

Fig 4.6 Plot of C_{Davg} , $C_{Lrms,}$ St vs Re for (a) Newtonian fluids (b) Non-Newtonian fluids

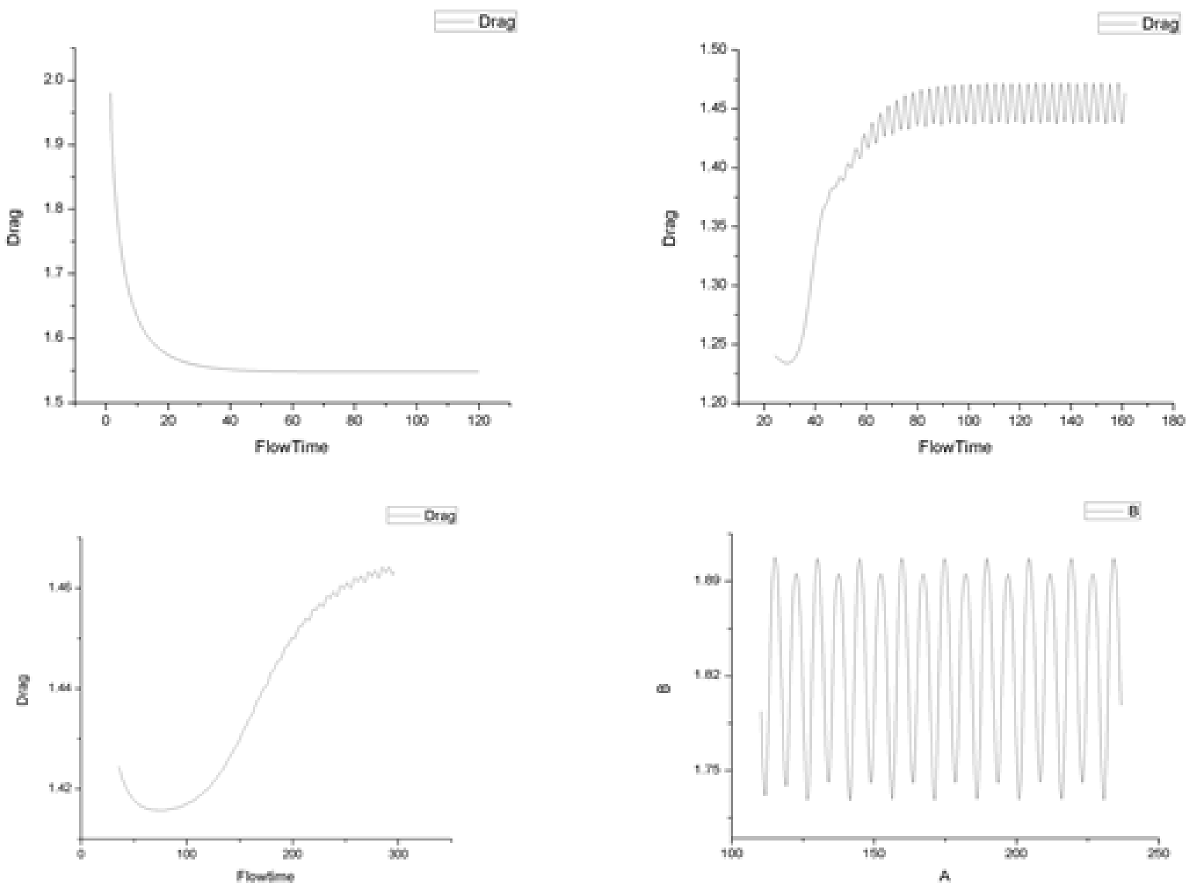

Fig 4.7 Time Dependence of Drag for Re 60 and 150 (a)Newtonian (b) Non-Newtonian

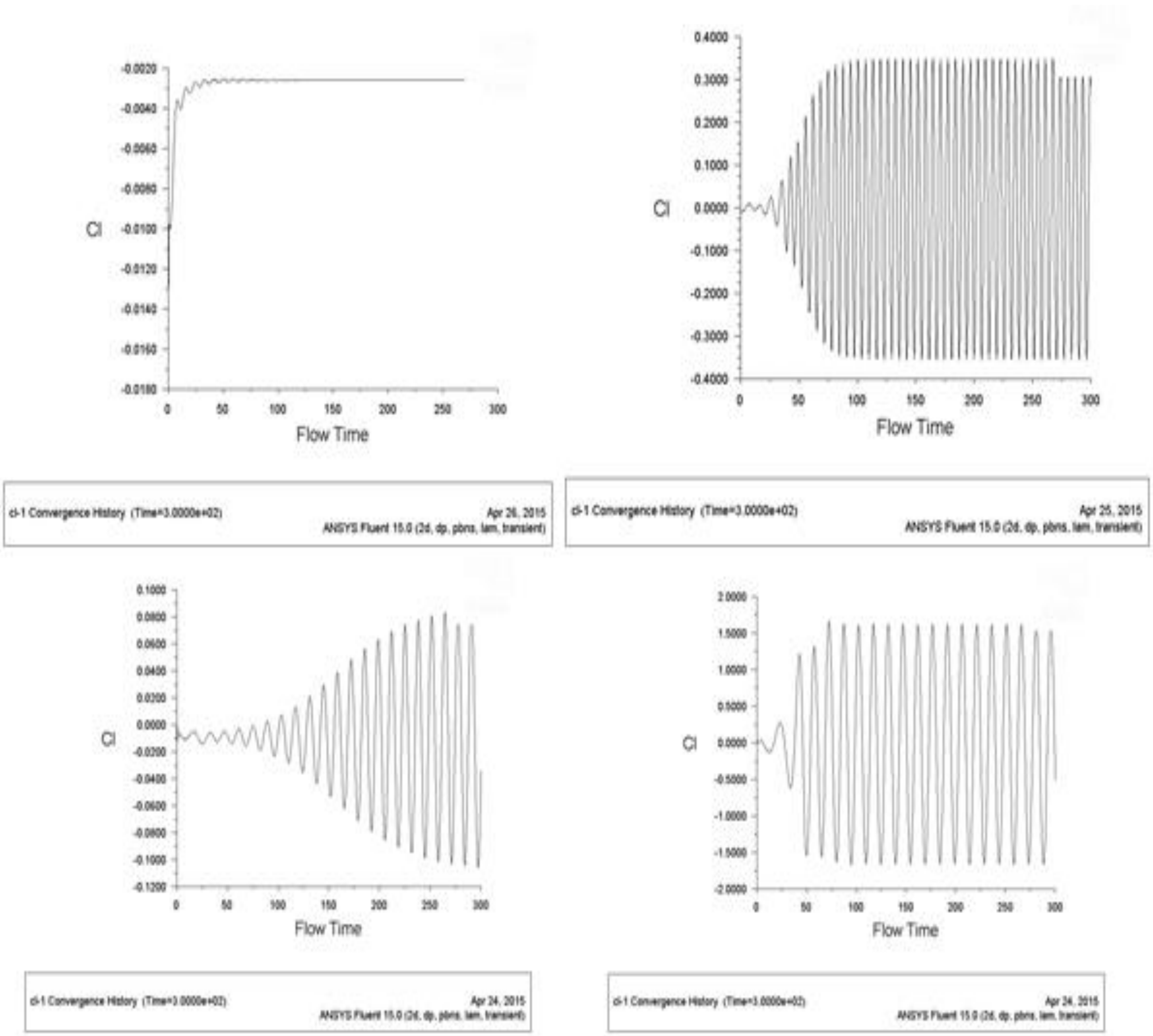

Fig 4.8 Time dependency of Lift for Re 60 and 150 (a) Newtonian (b) Non-Newtonian

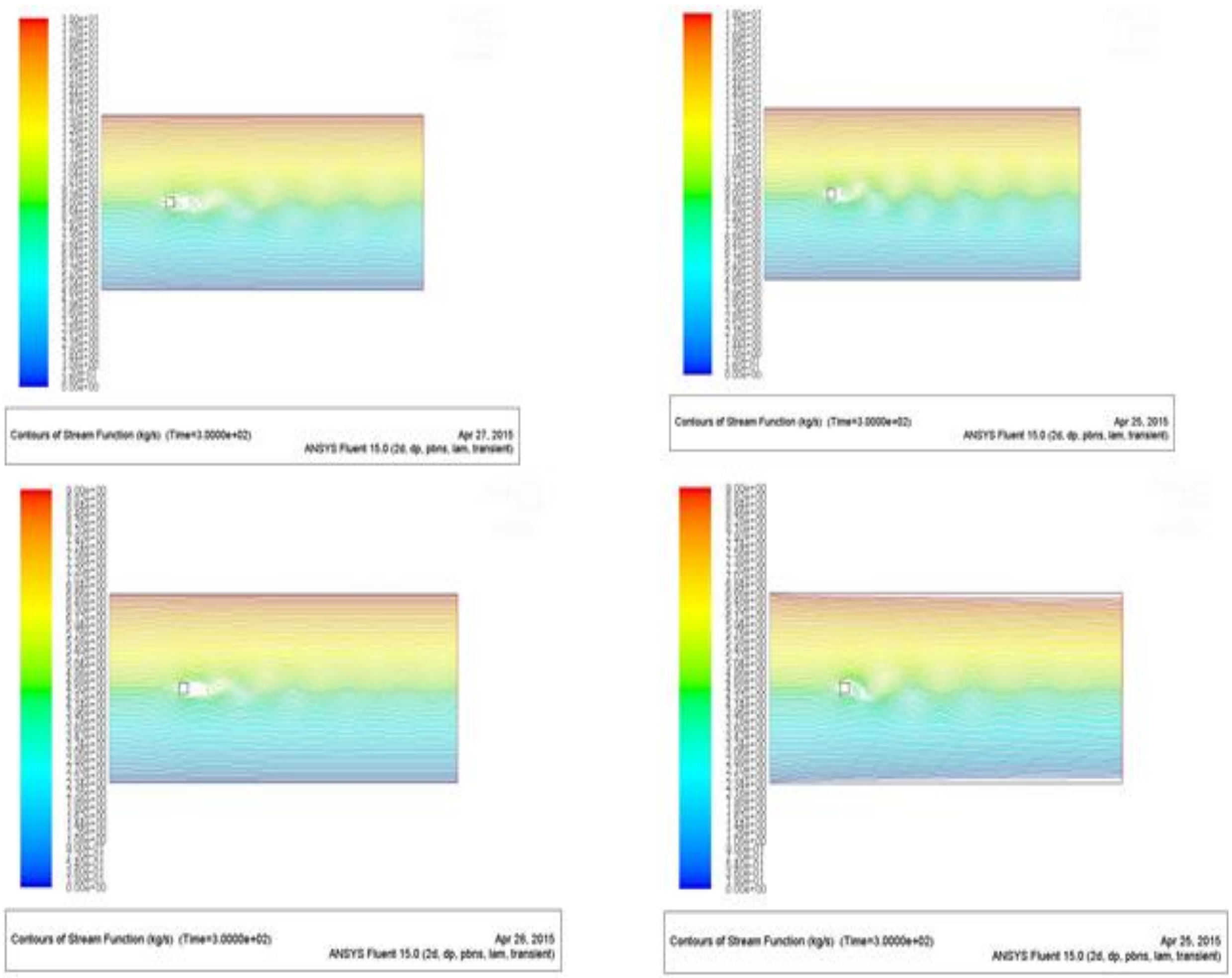

Fig 4.9 Instantaneous Streamlines for Re 60 and 150 (a) Newtonian and (b) Non-Newtonian

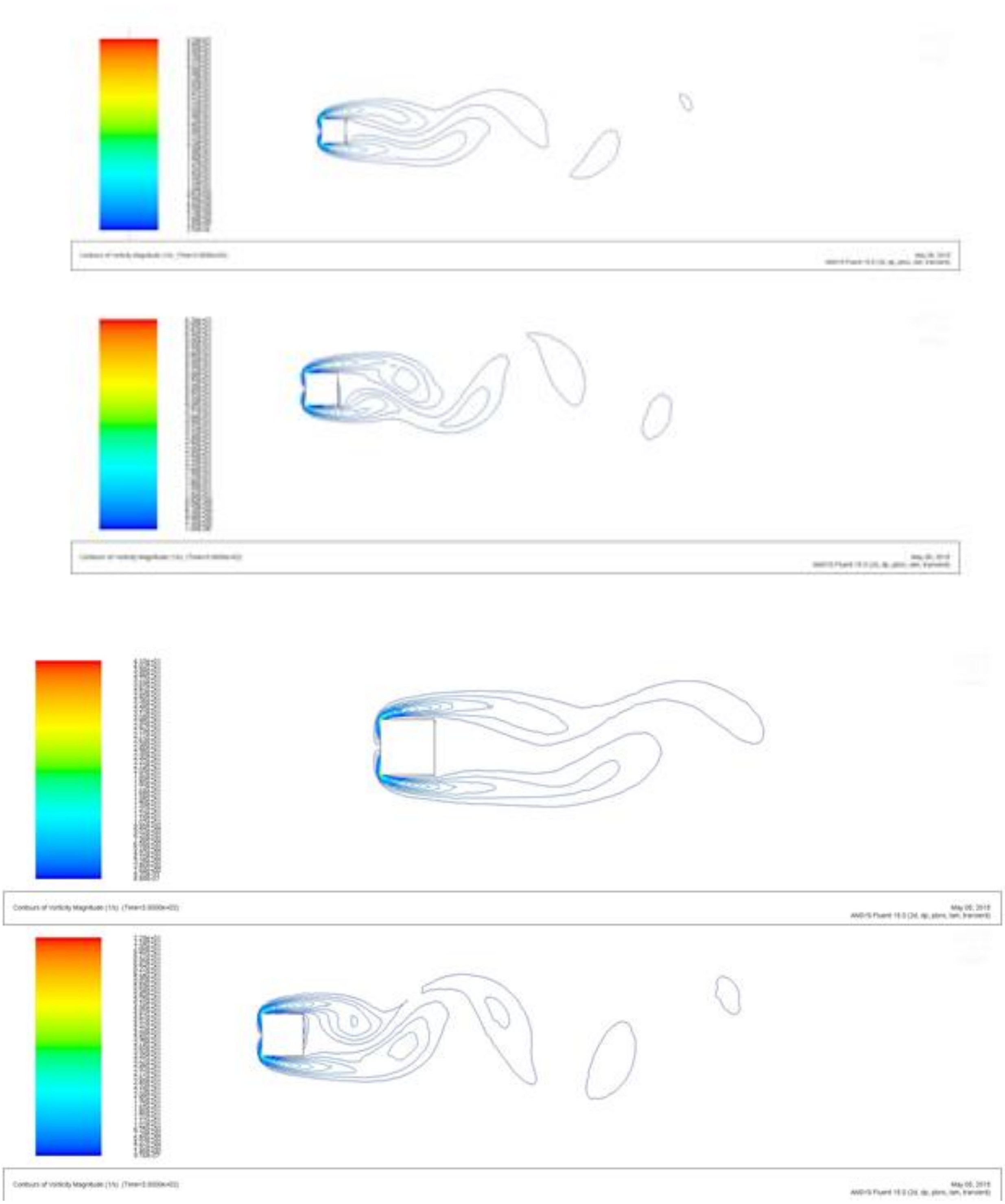

Fig 4.10 Vorticity Contours for Re 60 and 120 (a) Newtonian (b) Non-Newtonian

Chapter 5

Conclusions

For Non-Newtonian fluids, simulations were carried out for $60 \leq Re \leq 150$ and $0.5<n<1$ to show the effect of shear thinning behaviour on the fluid flow. Fig 5.4 (34-37) shows the instantaneous streamlines and vorticity magnitude for both unsteady Newtonian and non-newtonian fluids (n=0.5 and n=1). It shows the vortex shedding phenomenon (Fig5.4.42,43) where vortex formed at the rear part grows and detaches and this trend is responsible for the periodic flow. The results are in agreement to Sharma and Eswaran [6].

Depending on the value of n, the separation of leading edge starts for Re<100 for shear thinning fluids. With increase in power index, n, the effective viscosity of fluid increases and thus it stabilizes the fluid flow. In fluids with n<1 there is separate wake formation because of leading edge separation in addition to the one formed at the end part of the cylinder. With increase in Re, the wakes merge.

The Strouhal number value increases to a maximum and is then followed by a sharp drop. The reason is due to the edge or flow separation and lowering frequency of shedding. The value of average drag coefficients increase with the increase in Re. The C_{Lrms} basically represents the variations or oscillations amplitude. It increases with Re. The comparison with other results is shown in fig 5.4.27.

For Newtonian fluids, simulations were carried out at for $60 \leq Re \leq 150$ based on the literature parameters. Strouhal number, average drag coefficient and rms value of lift were calculated . Upto Re=50 the flow is steady and on increasing Re the flow becomes unstable and vortex shedding occurs in agreement to Sohankar et al[9]. The drag coefficients initially undergoes a sharp drop and then becomes constant. At higher value of Re, it leads to the phase of vortex shedding. Fig 5.4.25 for present work and comparison with other published results.

6. References

[1] K.N.Ghia, U Ghia, C.T.Shin, High-Re solutions for incompressible flow using the Navier Stokes equations and a multi-grid method, Journal of Computational Physics, 48(1982)387-411.

[2] Panagiotis Neofytou, A third order upwind finite volume method for generalized Newton fluid flows, Advances in Engineering software, 36(2005)664-680.

[3] R.P.Chhabra, A.K.Sahu, C.Sasmal, P.Koteswara Rao,V.Eswaran ,Effect of power law fluid behaviour on momentum and heat transfer characteristics of an inclined square cylinder in steady flow regime, International Journal of Heat and Mass Transfer, 54(2011)2854-2867.

[4] A.K. Dhiman, R.P. Chhabra, V. Eswaran, Flow and heat transfer across a confined square cylinder in the steady flow regime: effect of Peclet number, Int. J. Heat Mass Transfer 48 (2005) 4598–4614.

[5] B. Paliwal, R.P. Chhabra, A.Sharma, V. Eswaran, Power law fluid flow past a square cylinder: momentum and heat transfer characteristics, Chem. Eng. Sci. 58 (2003) 5315–5329.

[6] A. Sharma, V. Eswaran, Heat and fluid flow across a square cylinder in the two dimensional laminar flow regime, Numer. Heat Transfer A – Appl. 45 (2004) 247–269.

[7] M. Breuer, J. Bernsdorf, T. Zeiser, F. Durst, Accurate computations of the laminar flow past a square cylinder based on two different methods: lattice-Boltzmann and finite-volume, Int. J. Heat Fluid Flow 21 (2000) 186–196.

[8] S.Turki, H. Abbassi, S.B. Nasrallah, Effect of the blockage ratio on the flow in a channel with a built-in square cylinder, Comput. Mech. 33 (2003) 22–29.

[9] A. Sohankar, C. Norberg, L. Davidson, Low-Reynolds number flow around a cylinder at incidence: study of blockage, onset of vortex shedding and outlet boundary condition, Int. J. Numer. Methods Fluids 26 (1998) 39–56.
[10] A. Okajima, T. Nagahisa, A. Rokugoh, A numerical analysis of flow around rectangular cylinders, JSME Int. J. Ser. II – Fluids 33 (1990) 702–711.
[11] R.W. Davis, E.F. Moore, A numerical study of vortex shedding from rectangles, J. Fluid Mech. 116 (1982) 475–506.
[12] R.W. Davis, E.F. Moore, L.P. Purtell, A numerical-experimental study of confined flow around rectangular cylinders, Phys. Fluids 27 (1984) 46–59.
[13] A. Okajima, Strouhal number of rectangular cylinders, J. Fluid Mech. 123 (1982) 379–398.
[14] P.K.Sharma, B.Gera, R.K.Singh, CFD analysis of 2D unsteady flow around a square cylinder, International Journal of Applied Engineering Research, 1(2010)603-610.
[15] A.K.Sahu, R.P Chhabra, V.Eswaran, Two dimensional unsteady laminar flow of a power law fluid across a square cylinder, J.Non-newtonian fluid mech, 160(2009) 157-167.
[16] Y.Saito, T.Soma, R.Sagawa, H.Aoki, Y.Matsushita, M.Daikoku, M. Shirota, T. Inamura, Comparison of solution algorithm for flow around a square cylinder, International Conference on CFD in minerals and Process industries,10(2012)1-7.