Simulation of Turbulent Flow over A Square Cylinder and Free Shear Flow

Bhaskar Singh Chauhan
Patel College of Science and Engineering, Affiliated with Rajiv Gandhi Proudyogiki Vishwavidyala Indore, Madhya Pradesh, India

Abstract— Insecure stream around feigns bodies is a territory of extraordinary research for researchers for quite a long while. Streams around structures, fireplaces are models where the liquid is moving. Air scattering of poisons around feigns bodies has heightened the need to comprehend wake conduct. The vortex shedding recurrence relies upon various parts of the stream field, for example, the end conditions, blockage proportion of the stream section. In the present work, a numerical re-enactment was done for stream past a square chamber to see the wake conduct. A two-dimensional shaky stream past a square chamber has been examined numerically for the Reynolds number (Re) considered 21400 with the goal that stream is laminar. The fundamental targets of this examination were to catch the highlights of streams past a square chamber in a space with the utilization of CFD. The limited volume technique has been utilized with an amazing network game plan. The incompressible SIMPLE calculation was utilized for the speed pressure coupling. The second-request discretization was utilized both for reality. Force law plot was utilized on a non- uniform network and for time discretization Crank Nicholson was utilized. A high-goals matrix has been utilized to dodge misleading motions and to keep the blunders inside points of confinement. The lift coefficient and speed segment in the wake locale were checked for a count of Strouhal number. The variety of Strouhal number with Reynolds number was found from.

Keywords: Reynolds number, laminar, highlights of streams, Strouhal number SIMPLE calculation

I. INTRODUCTION

In today's world most of the fluid related work are practically turbulent flow not laminar flow. Laminar flow is well-ordered and is characterized by fluid layers sliding over one another. The flow is well-ordered in the macroscopic sense despite the chaotic motion of macroscopic level. In laminar flow the stream lines are not in random motion so they never intersect each other. In turbulent flow the streamlines are irregular and random in motion so they intersect with each other. The irregularity is manifested through complex variations of velocity with space and time (fluctuations). The irregular motion is generated due to random fluctuations. It is postulated that the fluctuations inherently come from disturbances (such as, roughness of the solid surface) and the may be either damped out due to viscous damping or may grow by drawing energy from the free streamline.

In the course of the most recent 100 years, the stream around slim tube-shaped feigns bodies have been the subject of serious research, for the most part, infer able from the building importance of basic structure, stream actuated vibration and acoustic discharges. As of late, such examinations have gotten a lot of consideration because of expanding PC capacities, upgrades in trial estimation strategies. Most by far of these examinations have been completed for the stream around a roundabout chamber, while, from a building purpose of see, it is additionally important to consider stream around other feigns body shapes, for example, honed rectangular cross-sectional chambers

II. LITERATURE REVIEW

Any type flow is generally described by the Navier Stokes Equation, So We start off by writing down the classical Navier Stokes equations for incompressible, constant viscosity, Newtonian fluids on the form

$$D\begin{pmatrix}u\\v\\w\end{pmatrix} = \frac{d}{dt}\begin{pmatrix}u\\v\\w\end{pmatrix} + (u.\Delta)\begin{pmatrix}u\\v\\w\end{pmatrix} = -\frac{P\Delta}{P} + g + \mu\Delta^2\begin{pmatrix}u\\v\\w\end{pmatrix}$$

Where this notation highlights the fact that the Operators $\frac{d}{dt}(u.\Delta)$ and $(\mu\Delta^2)$ must be applied to each component of the velocity vector u. To derive the Reynolds Averaged Nevier –Stokes Equations (RANS) we assume that the dependent variables (Velocity and pressure) consist of a mean part $\bar{\phi}$ and a random fluctuating turbulent part $\dot{\phi}$ such that $\phi = \bar{\phi} + \grave{\phi}$ This is called Reynolds decomposition. Before we go on into the algebra, we should not some laws for averaging.

A. Laws for averaging

Let the mean of a time signal $\phi(t)$ can be defined as $\bar{\phi} = \frac{1}{T}\int_0^T \phi dt$ where the integral is taken over a time period which is considerably larger than the period of the turbulent fluctuations. From this definition we may define several algebraic rules for manipulation of turbulent signals. Let $f = \bar{f} + \acute{f}$ and $g = \bar{g} + \acute{g}$ are two turbulent signal than

1) $\bar{\grave{f}} = 0$
2) $\bar{\bar{f}} = \bar{f}$
3) $\overline{\bar{f}g} = \bar{f}\ \bar{g}$
4) $\overline{f'g} = 0$
5) $\overline{f + g} = \bar{f} + \bar{g}$
6) $\overline{\bar{f}.\bar{g}} = \bar{f}.\bar{g}$
7) $\overline{fg} = \bar{f}\bar{g} + \overline{f'g'}$
8) $\overline{\frac{df}{ds}} = \frac{d\bar{f}}{ds}$
9) $\overline{\int f ds} = \int \bar{f} ds$

B. Deriving RANS (Reynolds Averaged Navier Stokes) Equation:

The process of Reynolds averaging is now to insert$u = \bar{u} + u'$ and$P = \bar{P} + P'$ thereafter take the average of the whole equation term for term, utilizing the laws above. The result is that the transport equation for$\bar{u}$ involves additional terms called Reynolds stresses, that acts as additional unknowns in the equations, and needs proper modelling to find suitable solution. As we will see, these Reynolds stresses emerge from the non-linear convective acceleration term. We will now conduct the Reynolds averaging of the u-momentum equation, and then generalize the result. Inserting$u = \bar{u} + u'$ and$P = \bar{P} + P'$ into the u-momentum, and then averaging the equation term for term gives:

Time Derivative

$$\overline{\frac{d(\bar{u}+u')}{dt}} = \overline{\frac{d\bar{u}}{dt}} + \overline{\frac{du'}{dt}} = \overline{\frac{d\bar{u}}{dt}} + \overline{\frac{du'}{dt}} = \frac{d\bar{\bar{u}}}{dt} + \frac{d\overline{u'}}{dt} = \frac{d\bar{u}}{dt}$$

Pressure gradient

$$\overline{\frac{d(\bar{P}+P')}{dx}} = \overline{\frac{d\bar{P}}{dx}} + \overline{\frac{du'}{dx}} = \overline{\frac{d\bar{P}}{dx}} + \overline{\frac{dP'}{dx}} = \frac{d\bar{\bar{P}}}{dx} + \frac{d\overline{P'}}{dx} = \frac{d\bar{P}}{dx}$$

Viscous Diffusion

$$\overline{\nabla^2(\bar{u}+u')} = \overline{\nabla^2\bar{u} + \nabla^2 u'} = \nabla^2\bar{\bar{u}} + \nabla^2\overline{u'} = \nabla^2\bar{u}$$

Convective acceleration

Before we calculate the non-linear convective acceleration term, we should utilize the following identity

$$(u.\nabla)\emptyset = \nabla.(u\emptyset) - \emptyset\,(\nabla.u)$$

Where of course the term $\emptyset(\nabla.u) = 0$, for an incompressible fluid. Reynolds averaging gives

$$\overline{([\bar{u}+u'].\nabla)(\bar{u}+u')} = \overline{\nabla.[(\bar{u}+u')(\bar{u}+u')]}$$
$$= \overline{\frac{d}{dx}\big((\bar{u}+u')(\bar{u}+u')\big)}$$
$$+ \overline{\frac{d}{dy}\big((\bar{v}+v')(\bar{u}+u')\big)}$$
$$+ \overline{\frac{d}{dz}\big((\bar{w}+w')(\bar{u}+u')\big)}$$

Let us work out the first of these terms :

$$\overline{\frac{d}{dx}\big((\bar{u}+u)(\bar{u}+u)\big)} = \overline{\frac{d}{dx}(\bar{u}\bar{u} + 2\bar{u}u' + u'u')}$$
$$= \frac{d(\bar{u}\bar{u})}{dx} + 2\frac{d\overline{\bar{u}u'}}{dx} + \frac{d\overline{u'u'}}{dx}$$

Where the averaging of the terms$\overline{\bar{u}u'} = 0$ according to the averaging laws, but the averaging of this terms $\overline{u'u'} \neq 0$, and this is the cause of the Reynolds stresses. We may generalize this results into to yield.

$$\overline{\frac{d}{dx}\big((\bar{u}+u')(\bar{u}+u')\big)} = \frac{d\bar{u}^2}{dx} + \frac{d\overline{u'u'}}{dx}$$
$$\overline{\frac{d}{dy}\big((\bar{v}+v')(\bar{u}+u')\big)} = \frac{d(\bar{u}\bar{v})}{dy} + \frac{d\overline{u'v'}}{dy}$$
$$\overline{\frac{d}{dz}\big((\bar{w}+w')(\bar{u}+u')\big)} = \frac{d(\bar{u}\bar{w})}{dz} + \frac{d\overline{u'w'}}{dz}$$

Such that

$$\overline{\nabla.[(\bar{u}+u')(\bar{u}+u')]} = \nabla.(\bar{u}\,u) + \nabla.\overline{u'u'}$$

We are now in a position to write the full form of the u-momentum RANS equation which becomes

$$\frac{d\bar{u}}{dt} + (\bar{u}.\nabla)\bar{u} = \frac{1}{\rho}\frac{d\bar{P}}{dx} + \bar{g} + v\nabla^2\bar{u} + \nabla.\overline{u'u'}$$

This result may be generalized in 3D to form the complete RANS equations

$$\frac{D\bar{u}}{Dt} = \frac{d}{dt}\begin{pmatrix}\bar{u}\\ \bar{v}\\ \bar{w}\end{pmatrix} + (\bar{u}.\nabla)\begin{pmatrix}\bar{u}\\ \bar{v}\\ \bar{w}\end{pmatrix}$$
$$= \frac{-\nabla\bar{P}}{\rho} + \bar{g} + v\nabla^2\begin{pmatrix}\bar{u}\\ \bar{v}\\ \bar{w}\end{pmatrix}$$
$$+ \nabla.\begin{pmatrix}\overline{-u'u'} & \overline{-u'v'} & \overline{-u'w'}\\ \overline{-v'u'} & \overline{-v'v'} & \overline{-u'w'}\\ \overline{-w'u'} & \overline{-w'v'} & \overline{-w'w'}\end{pmatrix}$$

The last term of this equations is the divergence of the Reynolds stress tensor written in this form, it illustrates that for each of the momentum equations, the divergence operator should be applied to the row vector of the corresponding momentum equation. For example if we were to write out the v-momentum, we must apply the divergence operator to the 2nd row of the Reynolds stress tensor.

III. EXPERIMENTAL SETUP AND METHDOLOGY

The Navier Stokes equation govern the velocity and pressure of a fluid flow. In a turbulent flow, each of these quantities may be decomposed into a mean part and a fluctuating part. Averaging the equations gives the Reynolds Averaged Navier Stokes Equation, which govern the mean flow. However, the nonlinearity of the Navier Stokes equations means that the velocity fluctuations still appear in the RANS equation, in the nonlinear term is known as the Reynolds stress, It's effect on the mean flow is like that of a stress term, such as from pressure or viscosity. This is a "unknowns" tensor called Reynolds stress tensor. No matter how many manipulations we do, we shall always end up with more statistical unknowns than the equations relating them. This is the famous turbulence closure problem.

So Reynolds Stress terms needs to be modelled using functions containing empirical constants and information about the mean flow hence known as "Turbulence Modelling".

There are also other ways to simulate the turbulent flows are:

– DIRECT NUMERICAL SIMULATION
– LARGE-EDDY SIMULATION (LES)

A. *Direct Numerical Simulation (DNS)*

DNS simply means numerical solving of Navier Stokes and continuity equation. When dealing with turbulent flow one tries to resolve all turbulent phenomena at all length and time scales simply by numerical solving of Navier Stokes and continuity equation. For a successful simulation one typically needs to know what order to set space grid and time steps of adequate scales. This data can easily be acquired by applying Kolmogorov turbulence theory in advance. DNS is of great importance. As computers develop one gains the capability to simulate flows at ever higher and higher Reynolds number. Now a days results acquired by DNS are so good that one may consider them equivalent to data gained experimentally

"Direct numerical reproduction", regularly truncated DNS, is a term for the most part utilized with regards to Computational Fluid Dynamics (CFD). The term is utilized as a distinguisher to systems, for example, RANS ("Reynolds-found the middle value of Navier–Stokes" and LES ("Large Eddie Simulations"). The hypothesis of liquid mechanics depends on the three protection laws of mechanics, the preservation of mass, the equalization of force, and the preservation of vitality, together with constitutive laws, for example, stress-strain rate relations and conditions of state. The constitutive laws contrast between various applications. Standard suspicions for a gas, for example, air, for example with regards to streamlined features, is that it is a perfect gas and a Newtonian liquid with no mass consistency.

Doing DNS just includes the discretization of these conditions and an answer of the subsequent arithmetical arrangement of conditions with a PC. That's it. In any case, these conditions are exceptionally testing to unravel, because of the outrageous scale contrasts, both in existence, that show up in ordinary applications. Actually, PCs are not ground-breaking enough to have the option to precisely recreate with DNS the stream around, state, a quick-moving vehicle. Besides, PCs won't inside the not so distant have the option to complete DNS for such circumstances. Methods, for example, LES and RANS have in this manner been created so as to inexact the specific conditions and give estimated arrangements through spatial averaging at the littlest scales (LES) or averaging in time (RANS).

B. Large-Eddy Simulation (LES)

LES involves the solution of the 3D time-dependent Navier Stokes equations. It treats both large scale motion and small scale motion differently. It uses direct simulation of the large-scale motion and use of a sub-grid-scale (SGS) model for turbulence of scales smaller than the computational grid spacing. The advantages of LES arise from the fact that the large eddies, which are hard to model in a universal way, are simulated directly. The application of LES to practical flows has been limited because of the prohibitive expense at high Reynolds number and the difficulties in specifying initial and boundary conditions. 10 years back, it was conceivable to compose thorough audits of what had been achieved utilizing immediate and enormous vortex recreation of choppiness [1, 2]. Since at that point, re-enactment methods have been applied to numerous streams that a total survey is never again conceivable. These techniques are currently settled as a supplement to exploratory techniques. In large eddy simulation (LES), just the biggest sizes of movement are mimicked expressly and the little scales are displayed. LES is costly however much less so than DNS. At whatever point DNS is doable, it is the strategy for decision. Be that as it may, when DNS is excessively costly, LES is currently is a decent other option; without a doubt, it is regularly sufficiently economical to consider for chosen mechanical applications Right now, portray the techniques utilized in LES, concentrating on the distinctions with DNS, particularly the models utilized to speak to the little scales and numerical strategies that can be utilized in complex geometries. We will investigate what is possible with LES today. We start with certain comments on choppiness and expectation techniques.

We already mention why we need turbulence modelling. There are different types of turbulence modelling.

C. Turbulence Models

Turbulence models of various complexity have been developed, and with very few exceptions, they can be classified as Eddy-viscosity models or Reynolds- stress models.

In EDDY-VISCOSITY MODELS, the unknown correlations are assumed to be proportional to the spatial gradients of the quantity they are meant to transport.

1) Eddy-Viscosity Concept

Most turbulence models use the eddy-viscosity concept to determine the Reynolds stresses from:

$$-\rho\overline{u_i u_j} = \mu_T\left(\frac{dU_i}{dx_j}+\frac{dU_j}{dx_i}\right)-\rho\times KE\ \frac{2d_j}{3}$$

Where the eddy viscosity and KE is the turbulent kinetic energy. It is not a fluid property, it depends on the state of turbulence and must be determined by the turbulence model.

Where can be written as

$$\mu_T = C_\rho V_s L_s$$

Where C is an empirical constant, and V_s and L_s are turbulence velocity and length scales which characterize the large-scale turbulent motion.

Kinematic eddy viscosity or turbulent viscosity is given by

$$EMUT = \frac{\mu_T}{\rho}$$

D. Classification of Turbulence Models

Zero-Equation Models – V_s and L_s are calculated directly from the local mean flow quantities (e.g. Prandtl's mixing-length model). One-Equation Models – V_s is calculated from a suitable transport equation, usually the turbulent kinetic energy, KE, and the length scale, L_s, is prescribed empirically (e.g. Prandtl's K-L model).

1) One Equation Models of Turbulence

- We have seen that while the MLH is attractively simple. It has some serious weaknesses making it unreliable for modelling complex flows.
- We thus go on to explore strategies for removing some of the weaknesses while still keeping the model relatively simple.
- Recall that in obtaining the MLH we wrote

$$v_t = c_\mu k^{\frac{1}{2}} l \quad \text{So that}$$
$$\overline{uv} = -v_t\frac{dU}{dy} = -c_\mu k^{\frac{1}{2}} l\frac{dU}{dy}$$

Substituting this into the local equilibrium condition allowed us to obtain an algebraic expression for $\boldsymbol{k}$, that could be used in the above expression for $\boldsymbol{v_t}$

- Suppose instead, that we retained the expression , but instead of using local equilibrium to determine, this was found by solving a closed form of the turbulent kinetic energy equation (see in earlier lectures):

$$\frac{Dk}{Dt} = \mathrm{Diff}_k - \overline{uv}\frac{dU}{dy} - \varepsilon = \mathrm{Diff}_k - \overline{uv}\frac{dU}{dy} - k^{3/2}/l$$

- As before, we can prescribe a variation for the length scale, so ε is taken as $k^{\frac{3}{2}} l$. the only unknown quantity in equation (2) is ten the diffusion of k.
- The turbulent diffusion of k is usually modelled as a gradient transport process:

$$\frac{Dk}{Dt} = \frac{d}{dy}\left(\frac{v_t}{\sigma_k}\frac{dk}{dy}\right) + P_k - \frac{k^{3/2}}{l}$$

The turbulent viscosity is taken as $v_t = c_\mu k^{1/2}$ and is used is the eddy-viscosity formulation to estimate the Reynolds stresses $\overline{u_i u_j}$

- The algebraic prescription l of is usually analogous to that of the mixing length l_m
- However different near-wall viscous damping terms can be associated with ε and v_t.

$$l_e = 2.4y(1 - \exp(-Ady^+))$$
$$l_\mu = 2.4y(1 - \exp(-A\mu y^+))$$
With $A_d = 0.016$, $A_\mu = 0.235$

- One advantage of solving an equation for k is that the turbulence energy and thus the viscosity does not now have to vanish when $\frac{dU}{dy}$vanishes locally.
- In the example shown, P_k is non-zero either side of the velocity maximum. Although it will be zero at the peak, k will be non-zero as it will be diffused across this region.
- Some transport effects have thus been incorporated into the turbulence model.

E. Other One-Equation Models

- Although we have looked at a transport equation for, other one-equation models have been proposed. Where the transport equation is for some other variable.
- Amongst others, Barth (1990), Spalart & Almaras (1994) and Menter (1994) have all developed models that solve for v_t itself.
- There are differences between the various model details, but in general they are based on combination of the transport equations for k and ε (the latter to be discussed later). A typical example is the of Menter (1994):

$$\frac{Dv_t}{Dt} = c_1 v_t \sqrt{\left(\frac{dU_i}{dx_j} - \frac{dU_j}{dx_i}\right)^2} - c_2 \frac{v_t^2}{l^2} + \frac{d}{dx_j}\left[\frac{v_t}{\sigma_v}\frac{dv_t}{dx_j}\right]$$

- Note that in local equilibrium, this educes to

$$v_t = \frac{c_1}{c_2} l^2 \Omega = \frac{c_1}{c_2} l^2 \left|\frac{dU}{dy}\right|$$

In simple shear $\left(\text{Where}\Omega \equiv \sqrt{\left(\frac{dU_i}{dx_j} - \frac{dU_j}{dx_i}\right)^2}\right)$

- However, the model does still need a length scale to be prescribed – for the destruction term in the v_t transport equation.
- Two-Equation Models – V_s and L_s are both calculated from transport equations, usually Kinetic Energy and its dissipation rate Epsilon.

F. Two – Equation Models

- In complex flows it is impossible to guess the distribution of length scales. It may be difficult to define the distance to the wall, and the local level of l is affected by convection and diffusion processes.
- One approach to avoid having to prescribe , building on the modelling already adopted for k , is to obtain l from its own separate equation.
- Here we thus consider how to devise a transport equation for some length scale containing variable of the form $k^a l^b$. Since we will also be solving the k transport equation , this will enable us to obtain l.
- The most popular choice of second variable has been ε the dissipation rate of k ie $= 3/2$. $b = -1(since\ \varepsilon = k^{3/2}/l)$.
- Other choices of variable have been proposed eg. a length scale L, or frequency f. One alternative that has been fairly widely tested (at least in aeronautical flows) is $\omega \equiv {}^{\varepsilon}/_{k}$

G. Decaying Grid Turbulence

- A suitable value for $c_\varepsilon 2$ can be found by considering decaying grid turbulence. If a uniform stream is passed through a turbulence-generating grid, the turbulence energy k will decay downstream of the grid (Since there are no velocity gradients, hence $P_k = 0$).
- The rate at which k decays with distance downstream of the grid can be measured experimentally and written as $k = cx^{-n}$ for some positive constants c and.
- In decaying homogeneous grid turbulence, Terms 1 and 3 in equation (7) can be neglected from the ε equation.
- The and equations thus reduce to

$$U\frac{dk}{dx} = -\varepsilon \quad \text{or}$$

$$\varepsilon = -U\frac{d}{dx}(cx^{-n}) = cnUx^{-(n+1)} \quad (1)$$

$$U\frac{d\varepsilon}{dx} = -c_{\varepsilon 2} 2\frac{\varepsilon^2}{k} \quad (2)$$

- Equation (1) gives an expression for ε. substituting this, and the expression for k , into equation (2) leads to:

$$-U\,cn\,(n+1)x^{-(n+2)} = -c_{\varepsilon 2}\frac{c^2n^2U^2x^{-2(n+1)}}{cx^{-n}}$$

- After cancelling common factors, we get

$$(n+1) = nc_{\varepsilon 2} \text{ Or } c_{\varepsilon 2} = \frac{n+1}{n} \quad (3)$$

- From experiments the decay rate $n \approx 1.1$, giving $c_{\varepsilon 2} \approx 1.9$
- Thus taking $c_{\varepsilon 2} \approx 1.9$ in our model should ensure that it will return the correct rate of decay of turbulence in the absence of any generation.

H. Log-Law Region of an Equilibrium Boundary Layer

- In an earlier lecture we saw that in the fully turbulent, local equilibrium, region of a simple boundary layer we should have:

$$P_k = \varepsilon \quad |\overline{uv}| = u_r^2$$
$$|uv|/k = c_\mu^{1/2} \quad \text{so} \quad k = u_r^2 / c_\mu^{1/2}$$

And hence

$$\frac{U}{u_r} = \frac{1}{k}\log(Eyu_r/v)$$

$$\text{So } \frac{dU}{dy} = \frac{u_r}{ky} \quad \text{and} \quad v_t = kyu_r$$

Where the friction velocity $u_r \equiv (T_w/\rho)^{1/2}$

- As a second constraint we can try to ensure that the modelled ε equation will return the above results in a simple boundary layer.
- The convection term $D\varepsilon/Dt$ can be neglected, but diffusion cannot be ignored, since the variation of ε near the wall is as y^{-1} (a length scale increasing linearly with wall-distance implies$\varepsilon = k^{3/2}/(2.5y)$).
- The ε transport equation in this case then becomes

$$0 = \frac{P_k^2}{k}(c_{\varepsilon 1} - c_{\varepsilon 2}) + \frac{d}{dy}\left[\frac{v_t}{\sigma_\varepsilon}\frac{d(P_k)}{dy}\right]$$
$$= \frac{(\overline{uv})^2}{k}\left(\frac{dU}{dy}\right)^2 (c_{\varepsilon 1} - c_{\varepsilon 2})$$
$$+ \frac{d}{dy}\left[\frac{v_t}{\sigma_\varepsilon}\frac{d}{dy}\left(-\overline{uv}\frac{dU}{dy}\right)\right]$$

Substituting in the earlier expressions for $\overline{uv}$, k , dU/dy and v_t then leads to

$$0 = u_r^4 \frac{u_r^2}{k^2y^2}\frac{c_\mu^{1/2}}{u_r^2}(c_{\varepsilon 1} - c_{\varepsilon 2}) + \frac{d}{dy}\left[\frac{kyu_r}{\sigma_\varepsilon}\frac{d}{dy}\left(u_r^2\frac{u_r}{ky}\right)\right]$$
$$= \frac{u_r^4 c_\mu^{1/2}}{k^2y^2}(c_{\varepsilon 1} - c_{\varepsilon 2}) + \frac{d}{dy}\left[\frac{u_r^4 y}{\sigma_\varepsilon}\frac{1}{y^2}\right]$$
$$= \frac{u_r^4 c_\mu^{1/2}}{k^2y^2}(c_{\varepsilon 1} - c_{\varepsilon 2}) + \frac{u_r^4}{\sigma_\varepsilon y^2}$$

Cancelling u_r^4/y^2 leaves a relation between the model coefficients $c_{\varepsilon 1}$, $c_{\varepsilon 2}$ and σ_ε .

$$\frac{c_\mu^{1/2}(c_{\varepsilon 1} - c_{\varepsilon 2})}{k^2} = -\frac{1}{\sigma_\varepsilon}$$

We thus want to choose model coefficients that satisfy this relation, to ensure the model will return an appropriate logarithmic velocity profile when applied to a local equilibrium boundary layer.

The model will now return a length scale $k^{3/2} / \varepsilon$ that does increase linearly with wall distance in a local equilibrium boundary layer. However, in more complex flows it need not always return this same variation (whereas in the simpler models considered, we imposed such a variation regardless of local flow conditions).

I. Boundary Treatment

The equations written above are only valid for fully turbulent high Reynolds number flow, and are thus not valid near the wall. Therefore these equations are not integrated all the way to the walls. Instead we use one of two options:

- Patch a turbulent wall-law (a logarithmic velocity profile) onto the solution near the walls
- Add damping functions to the equations.

J. Wall law Treatment

Let be a velocity component at a node p closest to the wall. We then impose that the velocity at this node should follow the logarithmic wall law

$$\frac{\bar{u}_p}{v^*} = \frac{1}{k}\ln\left(\frac{v^* y_p}{V}\right) + B$$

$$K_p = \frac{v^{*2}}{\sqrt{C_\mu}}$$

$$\epsilon_p = \frac{v^{*3}}{ky_p}$$

Where v^* is the wall friction velocity $v^* = \left(\frac{T_w}{\rho}\right)^{1/2}$, $K \approx 0.40$ is the Von Karman Constant.

IV. RESULTS

A. Simulation due to the obstruction

After the simulation we get to see how the flow is occurring due to the obstruction of the square cylinder. First we see how the U velocity means the resultant of u component velocity and v component velocity is changing throughout the section.

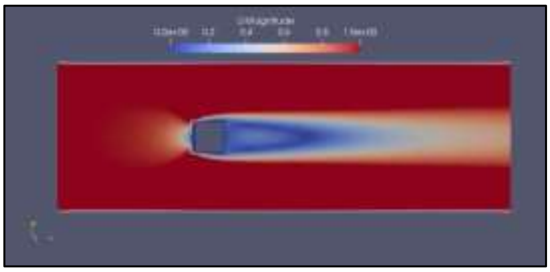


Fig. 4.1: simulation due to the obstruction

As you can see the U velocity is less after the obstruction and as we move further in x-direction it start to increase.

Now we see how the streamlines are behaving at different time of simulation: -

B. Under Developed Flow At Simulation Time Of 0.5 And 1 Seconds:

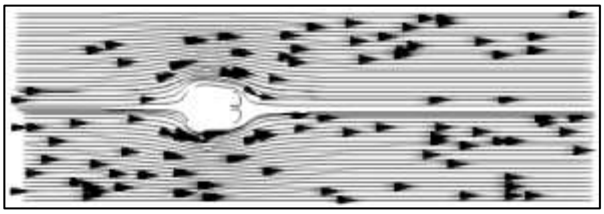


Fig 4.1 (a) streamline at time t=0.5

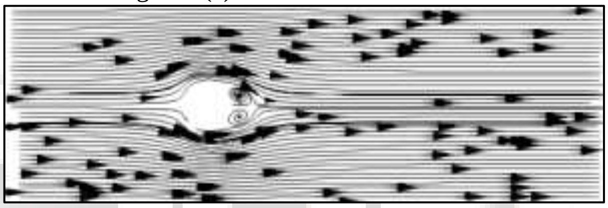


Fig 4.1 (b) streamline at time t=1

From the above figure shows us that flow is in under developed region and a large eddies are formed near the cylinder in downstream side in form the fig.1 we can say from downstream side velocity is low and also start squirreling so in this pressure drag will generate because cylinder upstream side stagnation point will occur near the square cylinder and at the stagnation point, stagnation pressure will be there which highest pressure across the flow. Due to this pressure difference pressure drag will occur across the square cylinder Stream lines at developing region at simulation time t =10 and 10.5 seconds:

Fig. 4.2 (a): Stream lines at developing region t =10

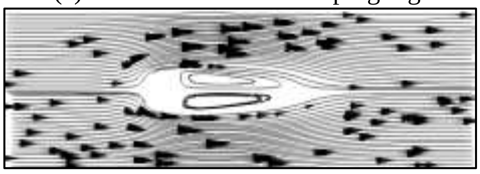


Fig. 4.2 (b): Stream lines at developing region t =15

At this time eddies has generated down-stream of the square cylinder and streamlines are very closed in middle of the cylinder at h\2 after wake region ended and eddies are also in stretched form.

C. Stream lines in fully developed region at simulation time 29.5,30 seconds.

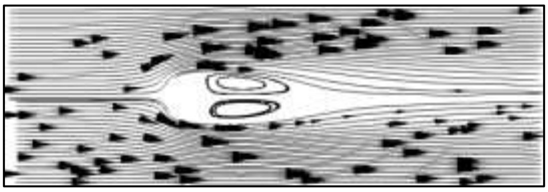


Fig. 4.3 (a): Stream lines developed region at simulation time 29.5

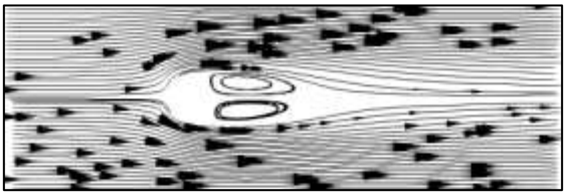


Fig. 4.3 (b): Stream lines developed region at simulation time 30

At this time effect of eddies reach to the other end of the cylinder because of this streamlines at h\2 in downstream region has some space. Eddies size also reduced as compare to developing region.

Apart from the variation of streamline we also see how kinetic energy and rate of dissipation varies throughout the section because of the obstruction provided by a square cylinder.

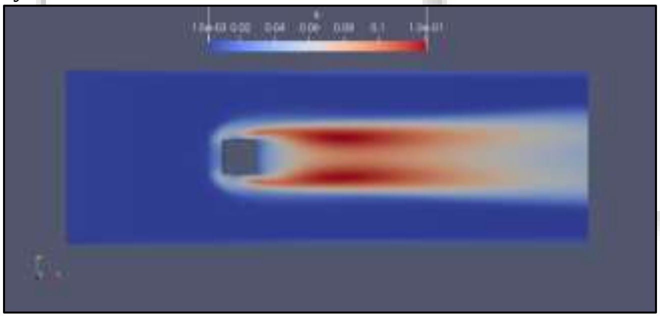


Fig. 4.4: kinetic energy distribution along x direction

From fig.4.4 we see that the kinetic energy/increases as we move further in x-direction but after certain distance the kinetic energy decrease as we move in x-direction.

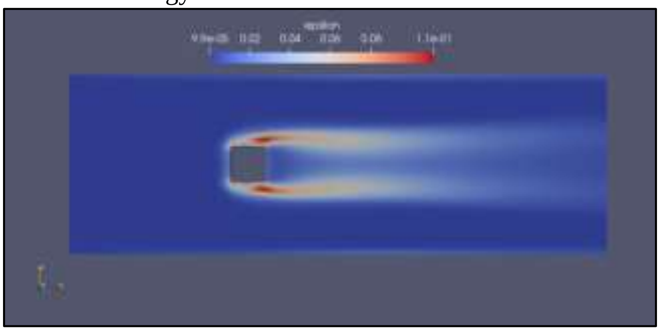


Fig. 4.5: dissipation rate varies throughout the section

From fig.4.5 we see how dissipation rate varies throughout the section. The rate of dissipation is high near at the top and bottom surface of the square cylinder and then its start to decrease as we move forward in x-direction.

We can also see how the pressure varies throughout the section from fig. 4.6

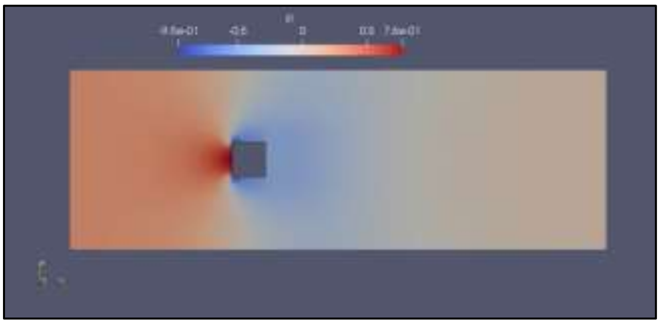


Fig. 4.6 Pressure Variation

D. Validation of results

For validation we refer to graphs from some research paper that are mention in reference section. First we compare the graphs between the u component velocity and the x-distance variation.

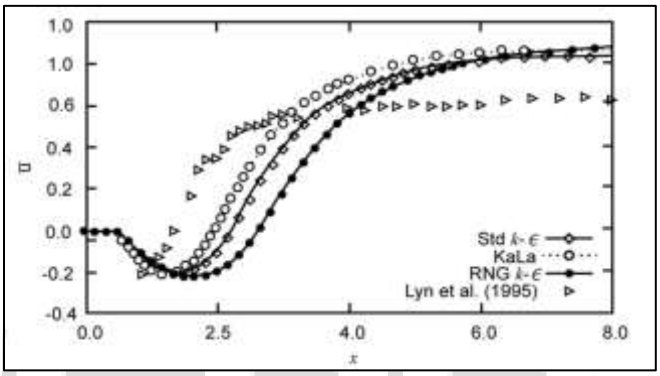


Fig. 4.7: Reference Graph

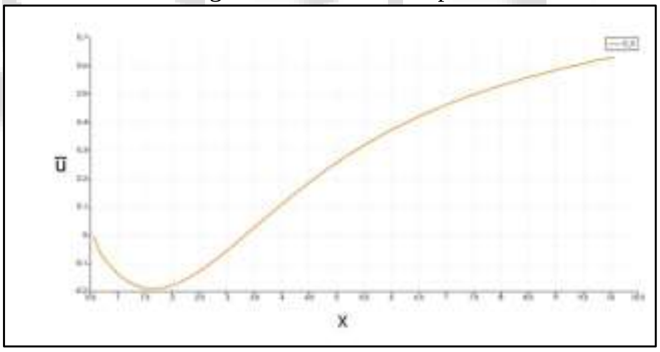


Fig. 4.8: Graph after simulation

The plot start from center of the square cylinder means there is x = 0, as we move in x direction first the velocity decrease and after some point it starts to increase. If we compare both the graphs we are getting that they are quite same. There are some deviations this is because the simulation will not give exact results it gives the idea how things happened.

Next graph comparison between the U velocity and the y-distance variation.

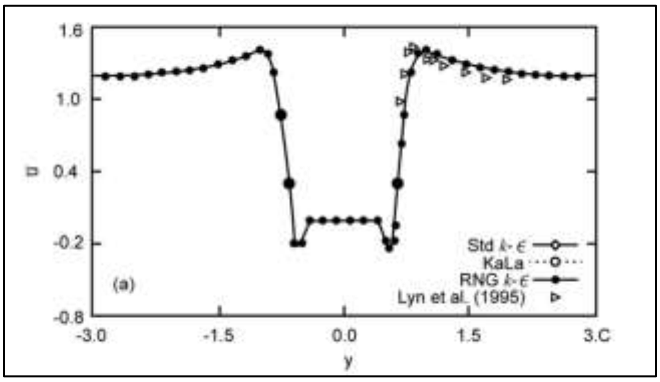


Fig. 4.9: second reference graph

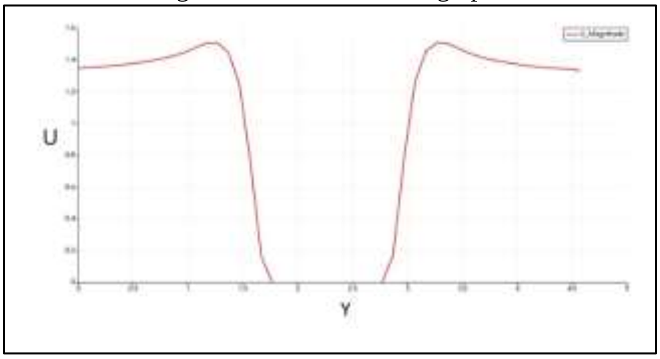


Fig. 4.10: Graph from simulation

As we see that both graphs look different is because the graph we get from research paper uses only u component velocity but our simulation graph uses U resultant velocity, so the variation is because of the v velocity component but still we see that the v velocity component does not affect the trend of the graph. In this graph y varies from bottom to top of the rectangle cavity which covers the square cylinder, u see in graph where velocity is zero is the square cylinder section. Also mention earlier that deviations is because there is some error in simulation.

Next graph comparison is between kinetic energy and the x-distance variation.

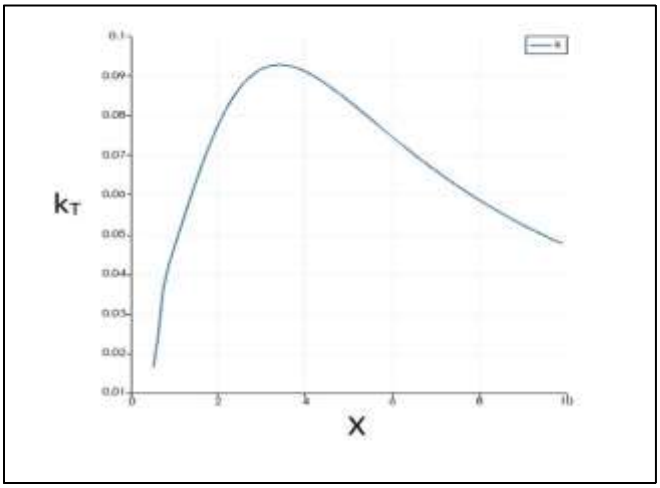
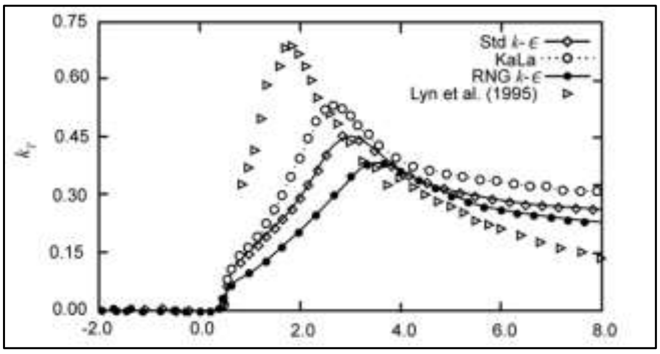


Fig. 4.11: graph we get from simulation



This graph is from the research paper

The graph we get from simulation and research paper follow the same trend like as we move forward in x-direction the kinetic energy starts to increase but after reaching its maximum value its starts to decrease. But in these graphs the deviation is quite high and the difference in the values of kinetic energy are also large. We think that there is some error that cause this deviation in the simulation part.From the above comparisons we say that our working model is no fully correct but still it gives the better understanding how the turbulent flow over a square cylinder. The properties we calculated like velocities, pressure, kinetic energy are good approx. Values from which we learn how this things changes throughout our rectangular section. This also gives the idea that the turbulence simulation is very tough because we have to deal with lot of unknown terms and then there is assumptions and no one knows what's really happened in the turbulence at micro.

V. SUMMARY AND CONCLUSIONS

Among the quantity of disturbance models in presence, the standard model is most mainstream and material to numerous unpredictable progressions of building significance. The model is computationally conservative and precision is sensible. Many enhancements for the model have been proposed in the ongoing past. The RNG and Kato-Launder are two such improved models. The overall execution of these models have been depicted together with the nitty gritty usage techniques.

The Phase-arrived at the midpoint of strategy prompts the standard RANS (Reynolds Averaged Navier Feeds) conditions with the exception of that the time-midpoints are supplanted by the gathering midpoints. Further, it depicts two sorts of variances viz., the vacillations due to the Periodicity of the mean stream and the other is the typical Stochastic changes. As such, the stage arrived at the midpoint of strategy is additionally named as shaky RANS, in which the complete range of the stochastic movement is mimicked by the choppiness model. The utilization of the Kato-Launder alteration on the standard condition has realized a significant improvement, which is affirmed through the presentation examination of various whirlpool consistency models. Various different adjustments have additionally been proposed to alter the vortex consistency models.

These are portrayed beneath. Speziale (1987) has proposed a non-straight model, adding nonlinear terms to the Boussinesq estimation of tempestuous anxieties. Speziale presented quadratic terms of mean speed inclinations and transport impacts of the strain tensor into the pressure strain relationship. This model predicts the auxiliary streams in a square conduit and yields more precise forecasts for isolated fierce streams past a regressive confronting step. Fundamentally, the nonlinear model conditions were created to represent the non isotropic conduct of the tempestuous anxieties, the tensorially invariant swirl consistency structure of the two-condition model was used.

Myong and Kasagi (1990) ans Shih et. al (1993) have additionally proposed quadratic nonlinear models. In spite of the fact that these models utilized comparable articulations in the pressure strain connection, the cofficients of their nonlinear terms are not the same as each other. Among these as of late proposed nonlinear models, the Myong and Kasagi (1990) model can be applied right totheir nonlinear terms are not quite the same as each other. Among these as of late proposed nonlinear models, the Myong and Kasagi (1990) model can be applied right to the divider; the various models are combined with the divider capacities. The model arrangements with the disturbance anisotropy and repeats an assortment of complex streams influenced by anisotropic Reynolds stresses. Durbin (1995) has demonstrated that the k-ϵ-(ν^2)$^{-}$ model can be applied effectively to complex isolated streams. In his model, (ν^2) is deciphered as a scalar, which handles precisely both the monstrous and smooth divisions.

VI. Conclusion

The graph we get from simulation and research paper follow the same trend like as we move forward in x-direction the kinetic energy starts to increase but after reaching its maximum value its starts to decrease. But in these graphs the deviation is quite high and the difference in the values of kinetic energy are also large. We think that there is some error that cause this deviation in the simulation part.From the above comparisons we say that our working model is no fully correct but still it gives the better understanding how the turbulent flow over a square cylinder. The properties we calculated like velocities, pressure, kinetic energy are good approx. Values from which we learn how this things changes throughout our rectangular section. This also gives the idea that the turbulence simulation is very tough because we have to deal with lot of unknown terms and then there is assumptions and no one knows what's really happened in the turbulence at micro.

VII. Scope for Future Research

The stream around slim tube-shaped feigns bodies have been the subject of serious research, for the most part, infer able from the building importance of basic structure, stream actuated vibration and acoustic discharges. As of late, such examinations have gotten a lot of consideration because of expanding PC capacities, upgrades in trial estimation strategies. Most by far of these examinations have been completed for the stream around a roundabout chamber, while, from a building purpose of see, it is additionally important to consider stream around other feigns body shapes, for example, honed rectangular cross•-sectional chambers.

References

[1] Lyn, D. A., Einav, S. , Rodi, W. And Park J. H., 1995, A Laser-Doppler Ve-locimetry Study of Ensemble-Averaged Characteristics of Turbulent Near Wake of a Square Cylinder, J. Fluid Mech. , Vol. 304, pp. 285-319.

[2] Myong, H. K. And Kasagi, N. , 1990, Prediction of Anisotropy of the Near-Well Turbulence with an Anisotropic Low-Reynolds Number K-e Turbulence Model, Journal of Fluids Engg (ASME) Vol. 112, pp 521-524

[3] The $k - \epsilon$ Model, the RNG $k - \epsilon$ Model and the Phase-Averaged Model-11 research paper by Gautam Biswas.

[4] Chen YS, Kim SW. Computation of turbulent flows using anextended k-epsilon turbulence closure model

[5] Kelkar, K. M.; and Patankar, S. V., Numerical prediction of vortex shedding behind asquare cylinder, International Journal Numerical Methods in Fluids vol (14), 1992, p.

[6] Sohankar, A.; Norberg, C.; and Davidson, L., Low•Reynolds•Number Flow around a Square Cylinder at Incidence: Study of Blockage, Onset of Vortex Shedding and Outlet Boundary Condition, International Journal for Numerical Methods in Fluidsvol (26), 1998, pp 39•56.

[7] Turki, S.; Abbassi, H.; and Nasrallah, S.B., Effect of the blockage ratio on the flow in a channel with a built•in square cylinder, Computational Mechanics, vol (33), 2003,

[8] Behr, M.; Liou, J.; Shih, R.; and Tezduyar, T. E., Vorticity stream function formulation of unsteady incompressible flow past a cylinder: sensitivity of the computed flow field to the location of the outflow boundary, International Journal Numerical Methods in Fluids vol (12), 1991, pp 323•342.

[9] Arnal, M. P.; Goering, D. J.; and Humphrey, J. A. C., Vortex shedding from a bluffbody adjacent to a plane sliding wall, Journal of Fluids Engineering vol (113), 1991,

[10] Shih, T. H., Zhu, J., and Lumley, 1993, A Realizable Reynolds-Stress Algebraic Equation Model, NASA Tm 105993.

[11] Myong, H. K. and Kasagi, N., 1990, Prediction of Anisotropy of the Near- Wall Turbulence with an Anisotropic Low-Reynolds Number *k- e* Turbulence Model, *Journal of Fluids Engg (ASME)* Vol. 112, pp 521-524

[12] Speziale, C. G., 1987, On Nonlinear *k-l* and *k-e* Models of Turbulence, J. Fluid Mech., Vol. 178, pp 450-475.

[13] Durbin, P. A., 1995, Separated Flow Computations with k-e-v2 Model, AIAAJ Vol. 33, pp. 659-664