CFD ANALYSIS OF AN AIRCRAFT AND FLOW OPTIMIZATION OF A PIPE UNDERGOING SUDDEN EXPANSION USING ANSYS

Zahid. F. Pathan[1]

[1]*Bachelor of Technology, Department of Mechanical Engineering, IIT Dharwad, Karnataka, India.*

---***---

Abstract - *CFD is a widely used numerical technique that deals with engineering problems that involves fluid flow. ANSYS is one of the most common CAE softwares used for CFD analysis. It has various methods of solving a problem. In this paper, we intend to perform and analyze two simulations. In SIMULATION 1, we perform an aerodynamic analysis of an aircraft at given boundary conditions. From this we calculate the pressure distribution, velocity streamlines and other important aspects. This is to give an overview of CFD in ANSYS. In SIMULATION 2, we simulate the flow of fluid in a pipe that undergoes expansion. Here we focus on aspects like turbulent kinetic energy and eddy dissipation and calculate it. In order to reduce these factors, the design of the pipe is modified and we run the simulation again with the same boundary conditions and check for turbulence kinetic energy and eddy dissipation. By reducing the effect of these two parameters, the model is optimized successfully.*

Key Words: **CFD, ANSYS, Aerodynamics, Aircraft, Pipe expansion, Wake region.**

1. INTRODUCTION

Computational fluid dynamics (CFD) comes under the subject of fluid mechanics where we use computers to solve for real life fluid flow problems using numerical analysis and data structures. It's a numerical technique that uses CAE software's to calculate for complicated equations. CFD is a widely used method for research and engineering problems. Its applications come under a variety of fields mainly aerodynamics, aerospace analysis, engine and combustion analysis, heat transfer and many more. The CAE software that we will use is ANSYS. It's a well renowned software for FEA and CFD analysis. It can be used in the analysis of strength, elasticity, temperature distribution, fluid flow and other attributes. It uses the method of meshing. The software divides the model to be analyzed, into a number of elements and solves for each and every element. The sub-division of ANSYS that we will use is ANSYS FLUENT. This is specifically built for CFD analysis and deals with problems involving a fluid.

2. LITERATURE STUDY.

S. Sanker [1] studied the effect of drag on an automobile. His objective was to reduce the aerodynamic drag on the car in order to maximize performance and increasing its fuel efficiency. He optimized his original model by adding a duct at the roof of the sedan modelled car. He performed the simulation in ANSYS Fluent and created the 3D model in CATIA V5. Through ANSYS Fluent CFD analysis he calculated and compared the coefficient of drag and pressure difference at the frontal and rear end of the car, in both the models. Madhu B P [2] researched on the CFD analysis of convergent-divergent and contour nozzle using numerical technique. She used ANSYS Fluent to calculate parameters like Mach number, static pressure and shocks for conical and contour nozzle. Her results depicted higher exit velocity and higher degree of flow separation with contour nozzles compared to corresponding conical nozzles. Ahmed H. Osman [3] interpreted the CFD simulation of De Laval nozzle geometry and effect of gas pressure variation at the entrance. He varied the total pressure at the entrance of two models and observed the change in Mach number at the outlet. He designed and analyzed the models in ANSYS Fluent. He used k-epsilon viscous model to perform the simulation. P.B Dehkordi [4] presented the hydrodynamic behavior of highly viscous oil-water flow through a horizontal pipe undergoing sudden expansion. He used ANSYS Fluent 16.2 and CFD approach by means of volume of fluid (VOF). Characteristics like pressure gradient, average velocity etc. he extracted from the CFD study. Mr.G.B.Nimadge [5] investigated the steady and incompressible fluid flow through a T-junction. He performed practical experiment as well as a simulation in ANSYS Fluent to validate the results. He concluded that there is much lower loss of energy in the T joint if the inlet pipe is at an angle and not exactly at 90 degrees to the horizontal pipe.

SIMULATION I

I. OBJECTIVE:

To perform aerodynamic simulation on an aircraft shown in Fig 1.1. This will give us an overview on performing CFD analysis in ANSYS Fluent and familiarize us with its mesh and solver settings. From this simulation we will calculate and present the pressure distribution, velocity

streamlines, eddy viscosity, turbulence kinetic energy (TKE) and turbulent eddy dissipation.

Fig -1.1: Rendered CAD model of the aircraft

II. SOLID MODELLING:

The solid model for the plane was created in Autodesk fusion 360. It can be modeled in any CAD software. The main tools used while creating it were FORM, Extrusions, Revolves, Fillets etc. Editing FORM tool was extensively used to get a well-defined aerodynamic structure. The image was rendered in Fusion 360 as well. The main Dimensions of the plane are as follows: Wingspan = 31.09 meters (102 ft), Length= 30.48 meters (100 ft) from the cockpit nose till the rudder end.

III. ANSYS:

Once we open workbench, we choose Fluid flow (Fluent) Analysis. There are various solvers in Ansys for CFD like fluent, CFX, polyflow. Here we will be using Fluent, which is widely used. Next, in design modeler we import the STEP file of the model. We add an enclosure which will act as Inlet, Outlet and walls later on in the simulation.

IV. MESHING:

Meshing is a process of discretizing the body into small elements. It's the most important step in a FEA Analysis. The Software will solve for each element to converge to a particular solution. The more the number of elements or smaller the size of elements, better will be the accuracy of the results. Here, we generated the mesh with default settings of fluent solver with a mesh size = 13m. As we are using Ansys student version for this particular simulation, we are limiting the number of elements to less than 5,12,000. After generating the mesh, we achieved number of nodes = 91,890 and number of elements = 5,03,601. The mesh has a default target skewness of 0.9. Next, we add named selections to the Inlet, Outlet and Walls.

V. CFD SOLVER:

This is the most crucial part of the analysis. Here the model is given necessary boundary conditions and is solved then analyzed. This is the most time-consuming step of the analysis. Finer the mesh, more the elements, longer the time needed to calculate the results. The following steps are taken to step up the model and solve it.

1. Solver: We use Pressure-based solver that has absolute velocity formulation. We do steady state analysis.
2. Models: For Aerodynamic Analytical model, the type of viscous based model used is k-epsilon (2 equations). This is the most common model used in CFD for turbulent flow conditions. It is a two-equation model that gives a general description of turbulence by means of two transport equations (PDEs) as follows:

   [6] For turbulent kinetic energy k :

   $$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\epsilon - Y_M + S_k$$

   For dissipation ϵ :

   $$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(P_k + C_{3\epsilon}P_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon$$

3. Assigning materials: Fluid material is Air and solid material is Aluminum.
4. Boundary conditions: The most essential part of the setup.
   (i) Inlet: Type = velocity-inlet, Velocity Magnitude = 604 kmph (Average velocity of a private jets, best suited for our model).
   (ii) Outlet: Type = pressure-outlet.
   (iii) Walls: Type = wall, Stationary, no-slip.

5. Solution method:
   (i) Pressure velocity coupling = Coupled.
   (ii) Gradient = Least squares cell method.
   (iii) Pressure = Second order.
   (iv) Momentum = Second order upwind

After completion of all the above steps and setting up boundary conditions, we initialize in Hybrid method and calculate for sufficient number of iterations. More the number of iterations, more accurate will be the results.

VI. RESULTS:

1.Velocity Distribution: Fig 1.2 shows the velocity streamlines at the mid plane of the model. From Fig 1.2 we can interpret that the air velocity is high near the surface of the plane. To be specific, near the wings and tail velocity is as high as 799 km/hr. The tail creates an area of turbulence and low air density.
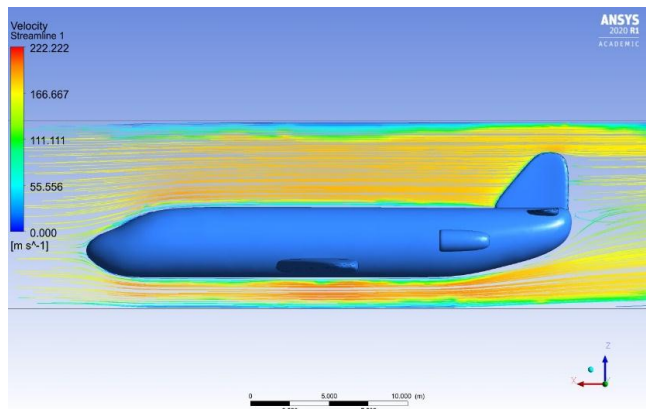
**Fig -1.2**: Velocity streamlines at the mid-plane.

Fig 1.3 Shows the Velocity vector at the wing, engine and horizonal stabilizer of the aircraft. The velocity is pretty high in these regions. At the wing, the velocity is around 710 to 750 km/hr. The velocity at the engine is about 740km/hr. The velocity at the bottom of the rear horizontal stabilizer is extremely high, about 800 km/hr.
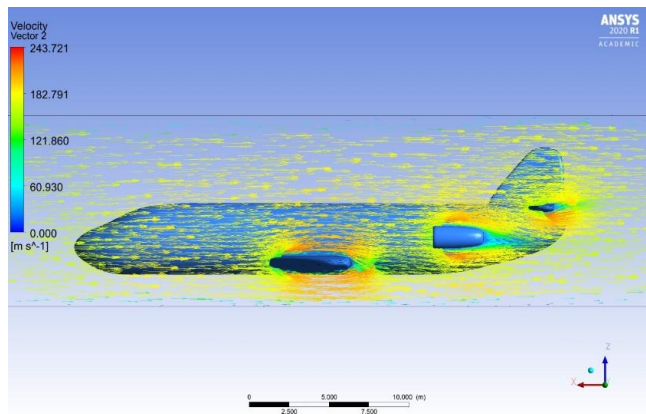
Fig - 1.3: Velocity vectors at the plane of wing, engine stabilizer.

2. *Pressure Distribution*: Fig.1.4 shows the pressure distribution on the mid plane of the aircraft. As we can see a high-pressure region at the nose or the cockpit of the aircraft. It is as high as around 5000 Pa. Fig. 1.5 shows pressure vector at the wings and horizontal stabilizer. These are high pressure areas. The pressure at the front of the wing is around 4300 to 5000 Pa. At the front of the horizontal stabilizer, pressure is around 3500 Pa There is a drastically

low pressure at the rare end of the aircraft. As the aircraft moves forward with high velocity, it creates a low-pressure region behind.
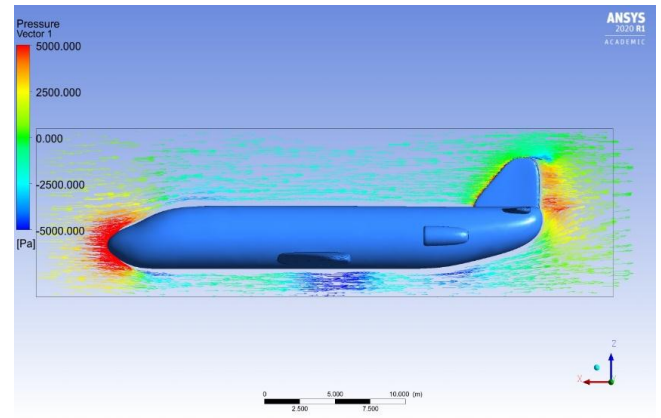

Fig – 1.4: Pressure distribution at the mid plane of aircraft.
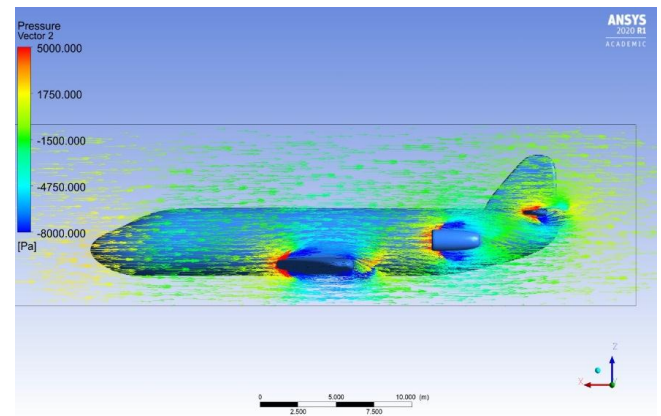
Fig -1.5: Pressure vectors at the plane of the wing.

3. *Eddy Viscosity*: Eddy viscosity is a coefficient relating average shear stress within a turbulent flow of air to the vertical gradient of velocity. It depends on fluid density and distance from ground surface. From Fig.1.6, We note that the eddy viscosity is around 0.155 Pa.s at the back of the wings. Its high at the rare of the engine, close to 0.5 Pa.s.
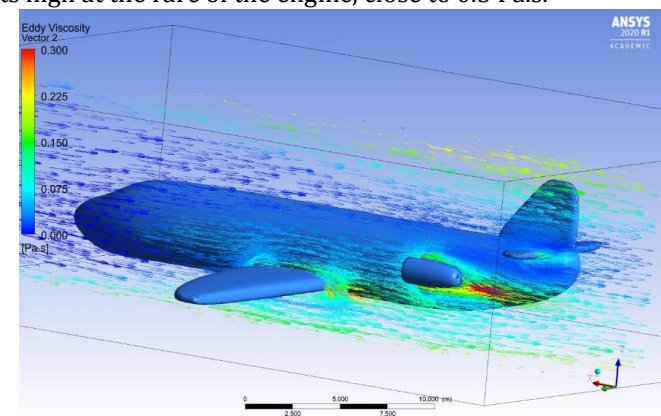
Fig – 1.6: Eddy viscosity distribution.

4. *Turbulence kinetic energy*: In fluid dynamics, turbulence kinetic energy (TKE, k in m²·s⁻²) is the mean kinetic energy per unit mass associated in turbulent flow. From Fig. 1.7 we can say that, It's comparatively high near the engines and k = 150 m²·s⁻² here.
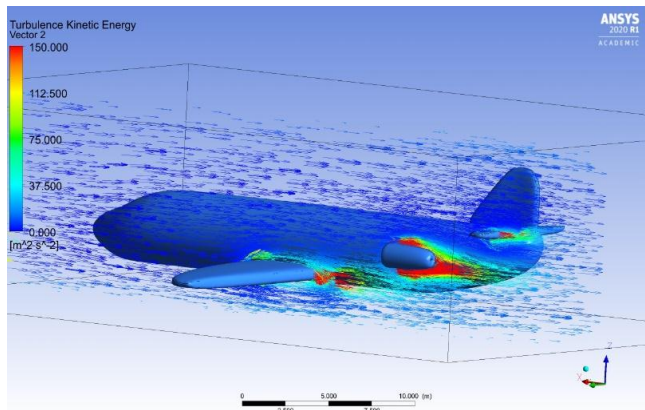

Fig -1.7: TKE Distribution at the mid plane of the engine.

5.*Turbulence eddy dissipation*:[7] Turbulent eddy dissipation rate (ε in m² s⁻³) represents the rate at which energy cascades from large to small eddies within the inertial subrange. This energy is eventually converted to thermal internal energy in the viscous subrange [Kolmogorov, 1941]. From Fig. 1.8 the Turbulence eddy dissipation is as high as 25000 m² s⁻³.
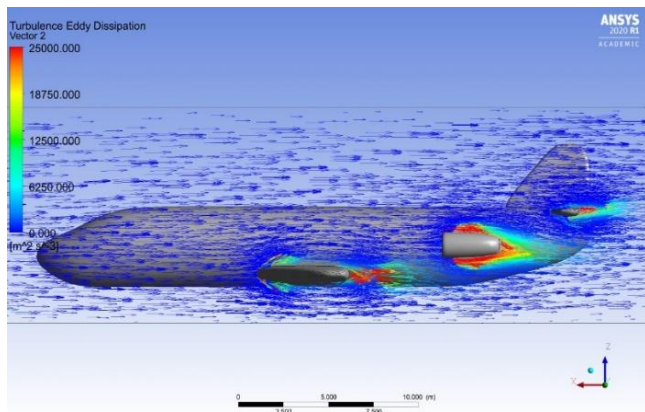

Fig -1.8: Turbulent eddy dissipation at wing and engine.

VII. CONCLUSION OF SIMULATION I:

From the obtained results, we conclude that we have successfully performed aerodynamic analysis on the given model of the air craft. We calculated and displayed some of the important parameters like velocity streamlines, pressure, eddy viscosity and information about turbulence. Therefore, we obtained an overall view in performing CFD analysis in ANSYS fluent.

SIMULATION II

I. OBJECTIVE:

In this CFD simulation, we are going to analyze the flow of a fluid through a pipe that undergoes sudden expansion. The fluid we consider here is water (liquid) and a Pipe of uniform circular cross section. This simulation has two parts. In PART A, we will consider expansion with sharp edged walls. We will calculate the turbulent eddy dissipation and TKE. Then, in PART B, we will modify our pipe design to optimize our results giving us much lower loss of energy through eddies. In both situations, the boundary and flow conditions will remain the same in order to make our comparison more effective.

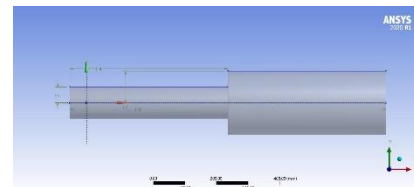**PART A:** Sharp-Edged Wall (Original Model).

Fig -2.1: Original design.

The Pipe dimensions are: Radius of small cross section = 50mm, Radius of large cross section = 100mm, total length of pipe = 1000mm. At the Expansion, the walls are sharp-edged (90°) as shown in Fig.2.1.

A1. MESHING:

After completing the Model in Design Modular, we move on to Ansys Mechanical to generate mesh. The mesh settings are shown in Table 2.1. After generating mesh successfully, we get a total number of 343,637 elements which is high enough for our model.

Mesh settings	
Meshing Method	Hex Dominant
Element size	0.007m
Refinement at area of expansion	Degree 1

Table -2.1: Mesh input settings

A2. CFD SOLVER:

Here, we will solve the model by giving exact boundary conditions, solid and fluid materials and by using

appropriate equations to get accurate results. Listed below are the steps we follow in CFD Solver.

1. General: We set Solver to Pressure based, Absolute velocity formulation and steady state analysis.
2. Models: Similar to the previous simulation, we use standard k-epsilon (2 equations), standard wall function based viscous model. In this model well can measure Turbulent Kinetic Energy (TKE) and turbulent eddy dissipations accurately.
3. Material and Cell zone conditions: We add Water (liquid) to the library of materials and set it to the cell zone condition
4. Boundary conditions: This is the crucial part of the setup.
   a. Inlet: Set type to Velocity inlet and velocity magnitude to 1m/s.
   b. Outlet: Type = outflow.
   c. Pipe wall: Stationary wall and no slip condition.
5. Solution method: We set the scheme to SIMPLEC. This algorithm is the modified version of the SIMPLE algorithm and is a commonly used numerical procedure in CFD to solve the Navier-Stokes equations. In spatial discretization, we set Gradient to green-gauss cell based. This will speed up our computation yet giving us satisfactory results. Pressure, momentum, TKE and turbulent dissipation rate are all set to Second Order Upwind to get much accurate results.

After completing all the above steps, we initialize our calculations from Input and Calculate for a sufficient number of iterations. More the number of iterations, closer will the results be to the actual values.
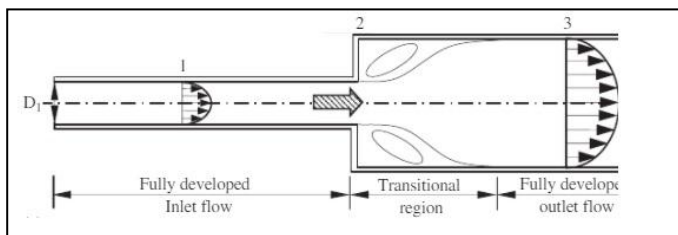

Fig -2.2: Three Main regions in the pipe.

A3. RESULTS OF PART A:

[8] Fig 2.2 shows a schematic of different regions in a pipe undergoing expansion. Its divided into 3 main zones. Fully developed inflow, transitional region and fully developed outflow. There is wake region at the walls of the transitional region. It contains eddy vortices.

1.*Turbulence Kinetic Energy*: Turbulence kinetic energy (TKE) is the mean kinetic energy per unit mass associated with eddies in turbulent flows. This and turbulent eddy dissipation are our main parameters of interest in this simulation. Fig 2.3 represents the TKE distribution at the mid plane of the pipe. We can clearly see the turbulence and the eddies in the translational region. In this region the maximum TKE is 0.048 J/kg or $m^2 \cdot s^{-2}$.
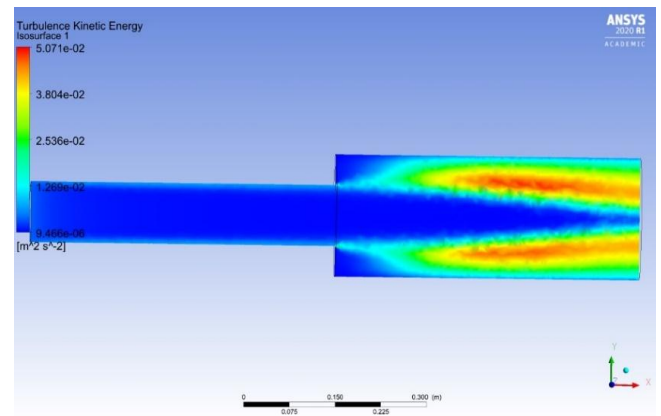

Fig - 2.3: Contour of TKE at the mid plane of the pipe.

2. Turbulent Eddy Dissipation: Fig 2.4 depicts the turbulent eddy dissipation at the pipe's mid plane. Its value is the highest at the sharp-edged walls at the beginning of the transitional region, and it is equal to 1.7 m^2/s^3. The value at the centre of the vortices is also comparatively high. Its magnitude is 0.25 m^2/s^3.
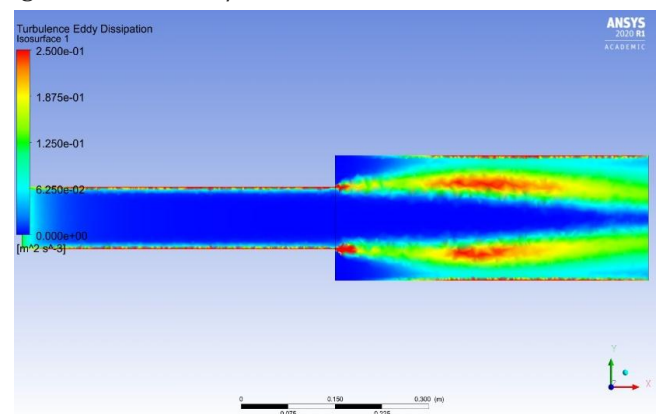

Fig – 2.4: Contour of turbulence eddy dissipation.

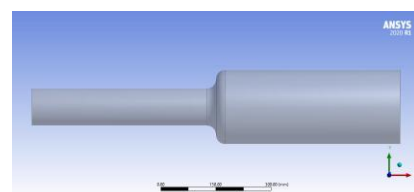**PART B:** Well-Rounded Wall (Modified model).

Fig - 2.5: Modified model with rounded walls.

Now, we modify our original model design by adding a fillet of radius 25mm at the expansion to get a well-rounded expansion wall. We keep all the other dimensions the same. The modified model is shown in Fig 2.5.

We follow the same procedure, settings and meshing methods as we did for the original model. Important note to set the same boundary conditions (inlet velocity = 1m/s) and solution methods in CFD Solver.

B1. RESULTS OF PART B:

1.Turbulence Kinetic Energy: Fig 2.6 shows the turbulence kinetic energy in the pipe. Its value is relatively high at the transitional region due to the formation of eddied. Its maximum value is 0.0389 J/kg as shown in red in the Fig. This value is lesser than the max value in the original model.
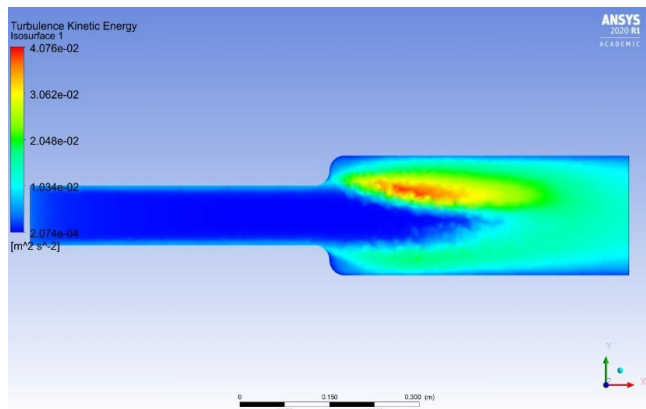

Fig -2.6: TKE in Modified Model.

2. Turbulence eddy dissipation: As in our modified model we have a well-rounded expansion region, the eddy dissipation is as low as 0.32 m^2/s^3 at the beginning of the transitional region. The value of eddy dissipation at the centre at the vortex is 0.2 m^2/s^3 which is lower to the original model. Fig 2.7 shows the distribution.
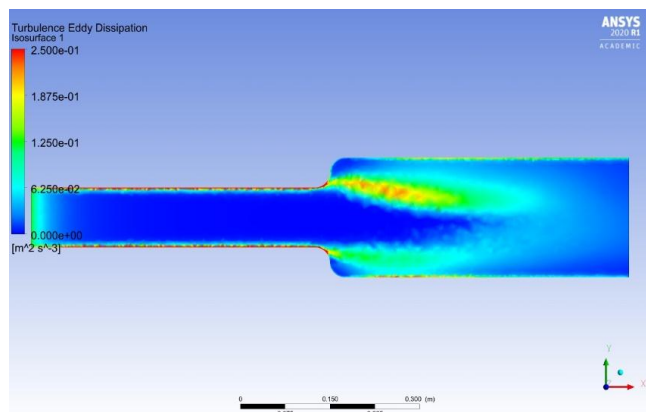

Fig -2.7: Turbulence eddy dissipation in modified model.

II. COMPARISON OF RESULTS IN PART A AND PART B:
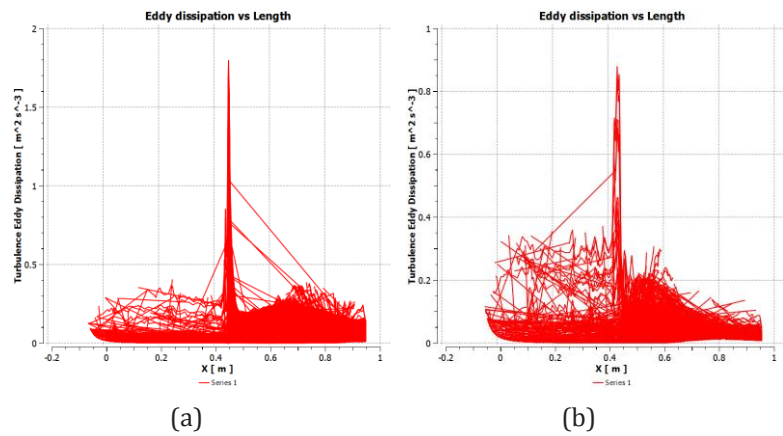

(a)   (b)

Chart-1: Turbulent eddy dissipation vs X in the original model (a) and Modified model (b) of the pipe.

The above figures show the Data interpretation in the form of charts. Chart-1 shows the graph of turbulent eddy dissipation vs the length of the pipe (x). Both the models have a sharp hike at x= 0.4m to x=0.5m but in the original model the highest value is around 1.7 m^2/s^3 and in the modified model its around 0.9 m^2/s^3.
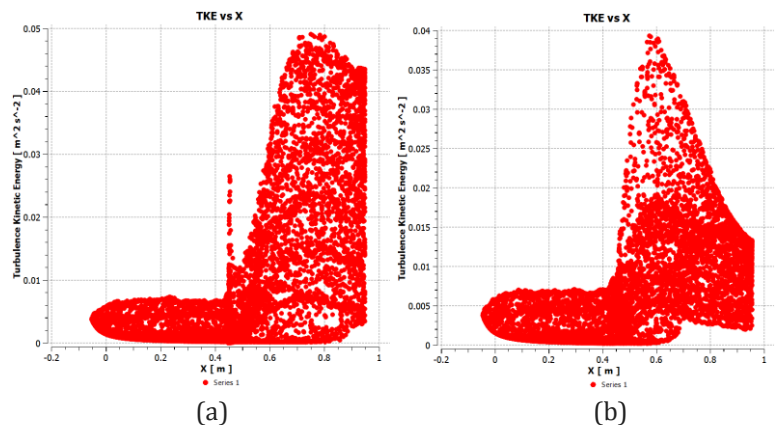

(a)   (b)

Chart-2: Turbulent Kinetic Energy vs X in the original model (a) and the modified model (b).

Chart -2 is plotted with TKE on the y axis and length of pipe (X) at the x axis. As it can be seen, the highest value in both differ. The highest value in (a) is around 0.05 $m^2 \cdot s^{-2}$ and in (b) around 0.04 $m^2 \cdot s^{-2}$. And also, in the modified model, the number of points with high TKE values are fewer as compared to the original model plot.

Fig 2.8 and Fig 2.9 show the velocity stream lines in the original model and Modified model respectively. There in less turbulence in the modified model.
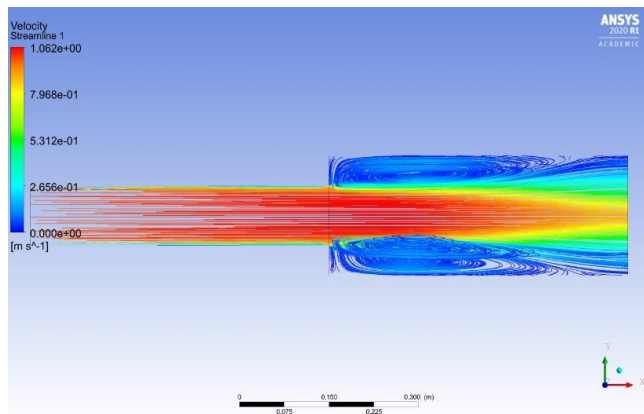
Fig -2.8: Velocity streamlines in original model.
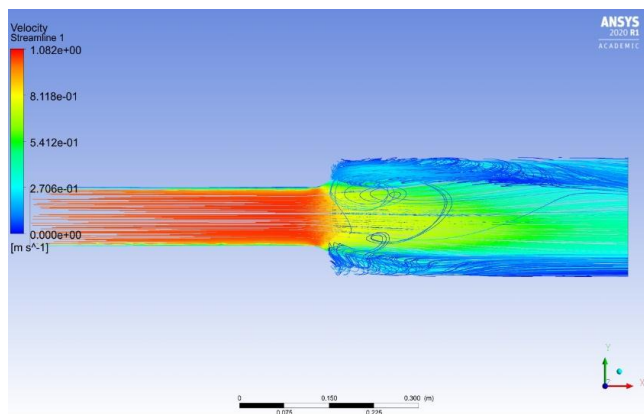
Fig -2.9: Velocity stream lines in modified model.

III. CONCLUSION:

From all the above results, figures and chart data we can conclude that our modified model has successfully reduced the turbulence kinetic energy and turbulent eddy dissipation in the pipe as compared to the original pipe model. We have minimized the energy lost due to turbulence. As a result, we will require lesser pumping power for the flow in the modified model as compared to the original one. Hence, we have successfully optimized the flow in the pipe by modifying the design of our pipe.

REFERENCES

[1] Dr. S. Sankar, CFD Analysis of Aerodynamics of Car, International Journal of Innovative Research in Science, Engineering and Technology, Vol.7, Issue 5, May 2018. ISSN (Online):2319-8753.

[2] Madhu B P and Syed Sameer, CFD Analysis of Convergent-Divergent and Contour Nozzle, International Journal of Mechanical Engineering and Technology, Vol.8, Issue 8, 2017. ISSN (Online):0976-6359.

[3] Ahmed H. Osman, CFD Analysis of De Laval Nozzle Geometry & Effect of Gas Pressure Variation at the Entrance, International Journal for Research in Applied Science & Engineering Technology, Vol.6, Issue XII, Dec 2018. ISSN: 2321-9653

[4] P. Babakhani Dehkordi, The hydrodynamic behavior of high viscous oil-water flow through horizontal pipe undergoing sudden expansion, Chemical Engineering Research and Design, Vol.139, Nov 2018, pages-144-161.

[5] Mr.G.B.Nimadge and Mr.S.V.Chopade, CFD Analysis of Flow through T-Junction of Pipe, International Research Journal of Engineering and Technology, Vol.4, Issue:2, Feb 2017. e-ISSN: 2395-0056.

[6] Versteeg, Henk Kaarle; Malalasekera, Weeratunge (2007). An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education.

[7] Paloma Borque, on the unified estimation of turbulence eddy dissipation rate using Doppler cloud Radars and lidars, Journal of Geophysical Research: Atmospheres, May 2016.

[8] Manmatha K. Roul and Sukanta K. Dash, Two-Phase pressure drop caused by sudden flow area contranction and expansion in small circular pipes, International Journal for Numerical Methods in Fluids, 66:1420-1446.

BIOGRAPHY

Zahid. F. Pathan
I am completing my under-graduation from IIT Dharwad, Karnataka, in the dept. of Mechanical Engineering. I have a zeal for solid modelling, FEA and CFD.