Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

CFD Analysis of Double Pipe Counter Flow Heat Exchanger

Dhrubajyoti Bhattacherjee
Department of Mechanical Engineering,
Swami Vivekananda Institute of Science & Technology.

***Abstract*:-The design of a double pipe in tube heat exchanger has been facing a problems because of the lack of experimental data available regarding the behaviour of fluid flow in double pipe & also in case of heat transfer data, which is not in the case in shell & tube heat exchanger. So,to the best of our effort double pipe heat exchanger by varying the different parameters like different temperatures & diameters of pipe & coil & also to determine the fluid flow pattern in a double pipe heat exchanger. The objective of this project is to obtain a better & more quantitative insight into the heat transfer process that occurs when a fluid flows. The study also covered the different types of fluid flow range extending from laminar flow through the transition to the turbulent flow. The materials for the study were decided, fluid taken was water and the material for the pipe was taken to be steel for its better conducting properties.**

Index term:-Double pipe Heat Exchanger, CFD Analysis, Solution, Results & Discussion, Conclusion, Report.

1.INTRODUCTION:-

Heat Exchange between flowing fluids is one of the most important physical process of concern, and a variety of heat exchangers are used in different types of installation, as in process industries, power plants ,food processing, refrigeration, etc. The purpose of constructing a heat exchanger is to get an efficient method of heat transfer from one fluid to another, by direct contact or by indirect contact. The heat transfer occurs by three principles: conduction, convection, or radiation. In a heat exchanger the heat transfer through the radiation is not taken into the account as it is negligible in comparison to conduction and convection. Conduction takes place when the heat from the higher temperature fluid flows through the surrounding solid wall. The conductive heat transfer can be maximized by selecting a minimum thickness of wall of a highly conductive material. But convection plays an important role in the performance of a heat exchanger.
Forced convection in a heat exchanger transfer t heat from one moving stream to another stream through the wall of the pipe. The cooler fluid removes heat from he hotter fluid as it flows along or cross it. Different co-relation are used for the calculation for the Nusselt no. and the heat transfer coefficient.

1.1.Double Pipe Heat Exchanger:-

A typical double pipe heat exchanger consists of one pipe placed concentrically inside another larger diameter with appropriate fittings to direct the flow from one section to next. As shown in fig.1 double pipe heat exchanger can be arranged into various series and parallel arrangements to meet pressure drop and mean temperature difference requirements. The major use of double pipe heat exchanger is for sensible heating and cooling of process fluid where small heat transfer areas is required (up to 50m^2).This configuration is also very suitable when one or both fluid are at high pressure. The major disadvantage is that double pipe heat exchanger are bulky and expensive per unit transfer surface inner tube may be single tube or multi tube. If the heat transfer coefficient is poor for annulus, axially finned inner tubes can be used. Generally double pipe heat exchanger are built in modular concept, i.e., in the form of hairpins. Study has been done on the type of the flows in the circular straight pipes, and the effect of Prandtl and Reynolds number on the flow patterns and on Nusselt numbers. The two basic boundary conditions that are faced in the applications are constant temperature and the constant heat flux of the wall.

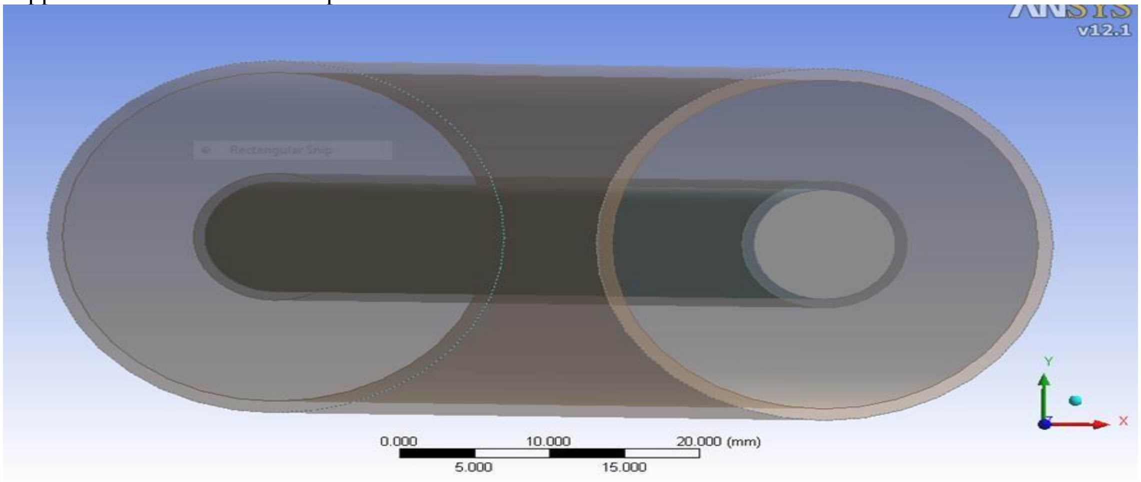


Figure 1.Double pipe heat Exchanger.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

1.2. ***Heat Transfer Co-efficient:-***

Convective heat transfer is the transfer of heat from one place to another by the movement of the fluids due to the difference in density across a film of the surrounding fluid over the hot surface. Through this film heat transfer takes place by thermal conduction and as thermal conductivity of most fluids is low, the main resistance lies there. Heat transfer through the film can be enhanced by in creasing the velocity of the fluid flowing over the surface which results in reduction in the thin film. The equation of the rate of heat transfer by convection under steady state is given by,

$$Q= h\ A\ (T_w-T_{atm})$$

Where,

h- is the film coefficent or surface coefficient ($W/m^2.K$).
A is the area of the wall.
T_w is the wall temperature.
T_{atm} is the surrounding temperature.

The value of "h" depends upon the properties of fluid within the film region; hence it is called 'Heat Transfer Co-efficient'. It depends on various properties of fluid, linear dimensions of surface and fluid velocity(i.e., nature of flow).

The overall heat transfer coefficient is the overall transfer rate of a series or parallel combination of convective and conductive walls. The 'overall Heat Transfer Coefficient' is expressed in terms of thermal resistances of each fluid stream. The summation of individual resistence is the total thermal resistance and its inverse is the overall heat transfer coefficient U.

$1/U = 1/h_o + (A_o/A_{\hat{\imath}}) \times (1/h_{\hat{\imath}}) + (A_o/A_{\hat{\imath}}) \times R_{fi} + R_w.$

Where,

U = overall heat transfer coefficient based on outside area of tube all
A = area of tube wall.
h = convective heat transfer coefficient.
R_f = thermal resistence due to fouling.
R_w = thermal resistence due to wall conduction

And suffixes "o" and "Î" refer to the outer and inner tubes, respectively.

1.3 Advantages of double pipe heat exchanger:-

- It is extremely easy to build and easy of operation. Its handle differential thermal expansions by the construction of the hairpin type component.
- The make use of longitudinal finned tubes, will having a low heat transfer coeffient.
- Easy to cleaning,maintainance,repair.
- Shortened delivery times can result from the use of stock components that can be assembled into standard sections.
- Modular design allows for the addition of sections at a later time or the rearrangement of the sections for new services.
- Simple construction leads to ease of cleaning inspection and tube element replacement.

1.4 Disadvantages of double pipe heat exchanger:-

- Hairpin Heat Exchanger sections are specifically intended units which are generally not built to any manufacturing industry standard other than ASME Code. However, TEMA tolerances are normally incorporated wherever applicable.
- Various hairpin heat exchanger section are not at all times cost-effectively competitive with a distinct shell and tube heat exchanger.
- Proprietary closure design requires special gaskets.

1.5. Double pipe heat exchanger applications:-

1. Pasteurization.
2. Digester
3. Heat Recovery
4. Pre-heating
5. Effluent cooling.

2.LITERATURE REVIEW:-

Heat Transfer enhancement in a heat exchanger is getting industrial importance because it gives the oppurtunity to reduce the heat transfer area for the heat exchanger.Increase in the heat exchanger performance can help to make energy, material and cost saving related to a heat exchange process.Double pipe heat exchangers are the simplest devices in which the heat is transferred from the hot fluid to the cold fluid through a seperating cylindrical wall. They are primarily adapted to high temperature and high pressure application due to their small diameters. They are fairly cheap, but the amount of space they occupy is relatively high comapared to the other types.Hence, for the given design and the length of the heat exchanger heat transfer enhancement in a double pipe heat exchanger is possibly achieved by several methods.Chen et al used dimples as the heat transfer modification on th inner tube.Bhuiya et al., Eiamsen et al. and Lioa et al. used circular tube equipped with perforated twisted tape inserts with the different configurations to enhance the heat transfer through the tubes.In order to intensify the heat transfer from the heat exchanger surface to fluids, it is possible to increase convection coefficient (by growing the fluid velocity), widens temperature difference between surface and fluids or increase the surface area acrross the which the convection occurs.Extended surfaces, in the form of

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

longitudinal or radical fins are common applications where the need to enhance the heat transfer between a surface and an adjacent fluid exists.Several researchers used extended surfaces for the enhancement in the heat transfer.

Masliyah et al. studied heat transfer characteristics for a lamiar forced convection fully developed flow in an internally traingular finned circular tube with axially uniform heat flux using a finite element method. For a given finned geometry, the nusselt number based on inside tube diameter was higher than that for a smooth tube.Also. it was found that for a maximum heat transfer there exists an optimum fin number for a given fin configuration.Agarwal et al. studied laminar flow and heat transfer magnitudes in a finned tube annulus.Pressure drop and heat transfer characteristics of the fins are obtained in the periodically fully developed region by varying geometric and flow parameters.Geometric parametrs are annulus radius ratio(0.3 to 0.5), fin height/annular gap(0.33 to 0.67) and fin spacing/annular gap (2 to 5).Flow parametrs are reynolds number(100 to 1000) and prandtl number (1 to 5).comparison are made with a plain tube annulus having the same length,heat transfer surafce area,volume flow rate and reynolds number.

They observed that a prandtl numbers less than 2,the use of fin may not be justified because the increase in pressure drop is more pronounced than the increase in heat transfer.At a Reynolds number of 1000 and a prandtl number of 5,the heat transfer increases by a factor of 3.1,while the pressure drop increases by a factor of 2.3.Solimann et al. studied steady, laminar, forced convection heat transfer in the thermal entrance region of internally finned tubes for the case of fully hydrodynamics.Results were presented for 16 geometries including the loacl nusselt number and developing length corresponding to each boundary condition.These results indicate that internal fining influences the thermal development in a complicated way which makes it inappropiate to extend the smooth tube results to internally finned tubes on a hydraulic diameter basis.Totala et al. conducted experiment in a double pipe heat excahnger by providing threads in the inner pipe.They observed that the nusselt number,heat transfer coefficient was increased for the threaded pipe. But the pumping power required also increased compared to the plain tube. Khannan et al. studied that the heat transfer through the double pipe heat excahnger with annular fins.Three different configurations annular ring, spiral rod and the rectangular projection were considered on the outside surface of the outer tube.Experiments were done with varied mass flow rates.It was observed that heat transfer rate was increased for a finned tube.Fin with annular ring showed better performance than the other methods.Nagarani et al. used circular and elliptical annular fins than the circular fins. Heat transfer coefficient depends on the fin spacing,flow condition and fluid properties.Fin efficiency was higher for elliptical fins. Mir et al. studied Numerical simulation of the steady, laminar, forced convection heat transfer in the finned anulus for the case of fully developed incompressible region of finned double pipe subjected to constant heat flux boundary conditions.They found a significant enhancement in heat transfer rate and also nusselt number.For small numbers of fins fin geometries like fin height,Ratio of Radii, and half fin angle were found to be less influential.

Iqbal et al. investigated optimal configuration of finned annulus with parabolic, traingular and trapezoidal fins using finite element methods and genetic algorithm.They concluded that no single fin shape is best in all situations and for all criteria.Zhang et al. studied heat transfer enhancement for shell side of a double-pipe heat exchanger with helical fins and pin fins, the three dimensional velocity components for the shell side with and without the pin fins were measured experimentally using the doppler anemometer (LDA) under cylindrical coordinate system and the fluid flow characteristics.The results show that,for the sheel side only the helical fins at large pitch,there was a pair of vortex near the upper and the lower edge of the rectangular cross sectionthe weakest secondary flow occurred at the center.By pin fins being installed,the three dimensional velocity components in the helical channel were strongly changed.Patel et al. simulated the industrial experimental of a double pipe heat exchanger using ANSYS14-CFX. Results indicated heat transfer,pressure drop,pumping power increased with mass flow rate whereas friction factor decreased.

For heat exchangers, built with many fins and designed for real industry,it is important to pay attention to and calculate the heat transfer considering the fluid flow and flow paths.The resistence of the body results in a pressure drop.Literature survey reveals that most of the analysis was done by considering constant heat flux or constant wall temperature boundary condition.Literature regarding the numerical study of enhancement in the heat transfer characteristics using the different configurations of the internal longitudinal fins for a double pipe heat exchanger with the conjugate heat transfer is still scare. Hence,the present work aimed at the comparison of heat transfer characteristics using the different mass flow rates and the temperature for parrallel and counter flow of a double pipe heat exchanger under various operating conditions to evolve with the best possible configurations.Numerical simulation was done using commercial CFD package. Heat transfer characteristics like temperature variations,heat transfer rate, and heat transfer coefficient for the above said models were compared and are presented.

3.INTRODUCTION TO CFD:-

Study of fluid flow has been around for millennium, dating back to ancient greece, but their understanding did not go beyond what they need to know to run aqueducts and other water works. Da Vinci further pursued the topic during the renainnace observing waves and free jets. Even newton studied fluids. The topic did not mature untill people like bernaulli and Euler investigated it and developed the equations that were named after them.

The Euler equations were further modified by Claude marry henry navier and george gabriel stokes to create navier stokes equation. These men laid the groundwork that would be the foundation of computational fluid dynamics.

Computational fluid dynamics is a term to describe a way of modelling the fluids using algorithms and numerical methods. Currently they are solved utilizing computers but early methods were completed manually without the aid of the computers. Computational fluid dynamics are a powerfull tool to model fluids, but even with the most state of an art supercomputers and technological advances they are only an approximation of what would occur in reality.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

It is unclear exactly when computational fluid dynamics came into being. Lewis Fry Richardson attempted to predict the weather by creating a grid in physical space and using Bjerkne " primitive diffential equation". His method involved a stadium of 64000 people each using a mechanical calculator to solve part of the flow equation.It ended in failure.
In 1933 A. Thom was able to numerically compute flow past a cylinder.Another mechanical solution was made by K.Kawaguiti which took 20 hours a week over 18 months. NASA s theoretical division also made contribution during 1960, but it wasnot untill the 1980s when commercial methods for computational fluid dynamics became available.
Fluid Flow is classified as classified as external and internal,depending on whether the fluid is forced to flow over a surface or, in a conduit. Fluid flow in circular and non-circular is commonly encountered in practise. The hot and cold water that we use in our homes is pumped through pipes. Water in a city is distributed by extensive piping network. Oil and natural gas are transported hundreds of miles by large pipelines. Blood is carried through our bodies by arteries and veins. The cooling water is an engine is transported by hoses to the pipes in radiator where it is cooled as it flows.
Thermal energy in a hydronic space heating system is transferred to the circulating water in the boiler and then it is transported to the desired location through the pipes.
So, it can be seen that many of the industrial processes have to deal with the flow throuh pipe. If analysis and simulation of flow through the pipe is done well than many of the industrial problems can be understood and solved using the results obtained.
Hence, in this project incompressible flow through the double pipe HEX, is simulated using CFD code software ANSYS FLUENT 17.1..

3.1 Computational Fluid Dynamics and Fluent:-

Computational fluid dynamics essentialy means computational transport phenomenon which involves fluid dynamics, heat and mass transfer or any phenomenon involving the transport phenomenon. CFD is about "numerical simulational of governing equations for some transport. Computational fluid is applied in many fields of science like aerospace, biomedical, and automobile,chemical industry, electronics, marine engineering, material processing, micro-fluids, turbo machines etc. Fluent is a CFD code used for flow modelling applications. Fluent needs some input data and domain draws from various software like GAMBIT,CATIA,PRO-E, and other design software. Now fluent can analyse the given domain with boundary conditions and solve the governing equations for the flow given the different parametrs.Steps for analysis in the fluent are:-
A)Pre-processing:- It involves building a model or importing one from a CAD package, applying.
B)Calculation:- Once the numerical model is prepared fluent performs the necessary calculation and produces the desired model.
C)Post-processing:- It involves organization and interpolation of the data and images.
It can handle:-

- Transient and steady flow
- Laminar and turbuent flow
- Newtonian and non-Newtonian Fluid
- Single and multiphase flow
- Chemical reactions including combination.
- Flows through porous media
- Heat transfer
- Flow induced vibration.

This project and Our research basically deals with turbulent flow of water through the double pipe. It also contains heat transfer and single phase flow.

3.2 Basic Principle That Govern The Implementation of CFD:-

Fundamental principles of conservation which governs the basic equation that we commonly used for CFD and not only CFD is analytical fluid dynamics are:-

- **Continuity equation:** A Mass Conservation Equation.

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad \text{(For 2D incompressible flow)}$$

- **Navier-stokes equation:** A Momentum Conservation Equation.

$$\frac{\partial u}{\partial t}+u\left(\frac{\partial u}{\partial x}\right)+v\left(\frac{\partial u}{\partial y}\right)=-\left(\frac{\partial \rho}{\partial x}\right)+\mu[\partial^2 u/\partial x^2+\partial^2 u/\partial y^2]$$

(for 2D incompressible flow in X-direction).

- **Energy Equation:** Energy Conservation Equation.

$$\partial T/\partial t+u\left(\frac{\partial T}{\partial x}\right)+v\left(\frac{\partial T}{\partial x}\right)=\frac{K}{\rho c}[\partial^2 T/\partial x^2+\partial^2 T/\partial y^2].$$

3.3.Major Key findings and gap analysis:-

The brief story of computational fluid dyanamics can be shown:-
Untill 1910: Improvements on mathematical models and numerical methods.
1910-1940: Integration of models and methods to generate numerical solutions based on hand calculations.
1940-1950: Transition to computer based calculations with early computers (ENIAC) solution for flow around cylinder by kawaguti with a mechanical desk calculator in 1953.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

1950-1960: Initial study using the computers to model fluid flow based on the navier stokes equation by los Alamas national lab,US. Evaluation of vorticity-stream function method. First implementation for 2D, transient, incompressible flow in the world.
1960-1970: First scientific paper "Calculation of potential flow about arbitary bodies" was published about the computational analysis of 3D bodies by Hess and Smith in 1967. Generation of commercial codes. Contribution of various methods such k-Ɛ turbulent model, Arbitary lagrangian-Eulerian, SIMPLE algorithm which are all still broadly used.
1970-1980: Codes generated by Boeing,NASA and some have unveiled and started to use several yeilds such as submarines,surface ships and automobiles,helicopters and aircrafts.
1980-1990: Improvement of accurate solution of transonic flows in three dimensional case by james et.al. Commercial codes have started to implement through both the academia and industry.
1990- Presesnt: Through the development in Informatics; worldwide usage of CFD virtually in every sector.

4.CFD ANALYSIS:-

Computational fluid dynamics (CFD) is the study of the system starts with the construction of desired geometry and mesh for modelling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the decretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modelling starts with the describing of the boundary and initial conditions for the dominion and leads to modelling of the entire system. Finally, it is followed by the analysis of the results, conclussions and discussions.

4.1.Geometry:-

Heat exchanger is built in the ANSYS workbench design module. It is a counter-flow heat exchanger. First, the fluid flow (fluent) module from the workbench is selected. The design modellar opens as a new window as the geometry is double-clicked.

4.1.1.sketching:-

Out of the three plane viz.,XY-plane, YZ-plane and ZX-plane, the XY-plane is selected for the first sketch.A circle of diameter 70mm & another circle with diameter 66mm is drawn by selecting the circle in sketching tab menu.Similarly by selecting again XY-plane we draw another sketch with two circle having diameter 42mm and 34mm.

4.1.2.Extrude:-

By selecting the first sketch in XY-plane extrude it up to a length of 1250mm name as a outer pipe.Again Select another sketch Extrude it upto 1400mm & name as inner pipe and change its operation from material to add frozen in Z-direction and change operation add material to add frozen.By extruding the whole body and by cahnging operation from add material to add frozen we have four different parts and four bodies.

Table 1. Naming of various parts of the body with the state type:-

Part Number	Part of the Model	State Type
1.	Inner fluid	Fluid (HOT WATER)
2.	Inner pipe	Solid (STEEL)
3.	Outer fluid	Fluid (COLD WATER)
4.	Outer pipe	Solid (STEEL)

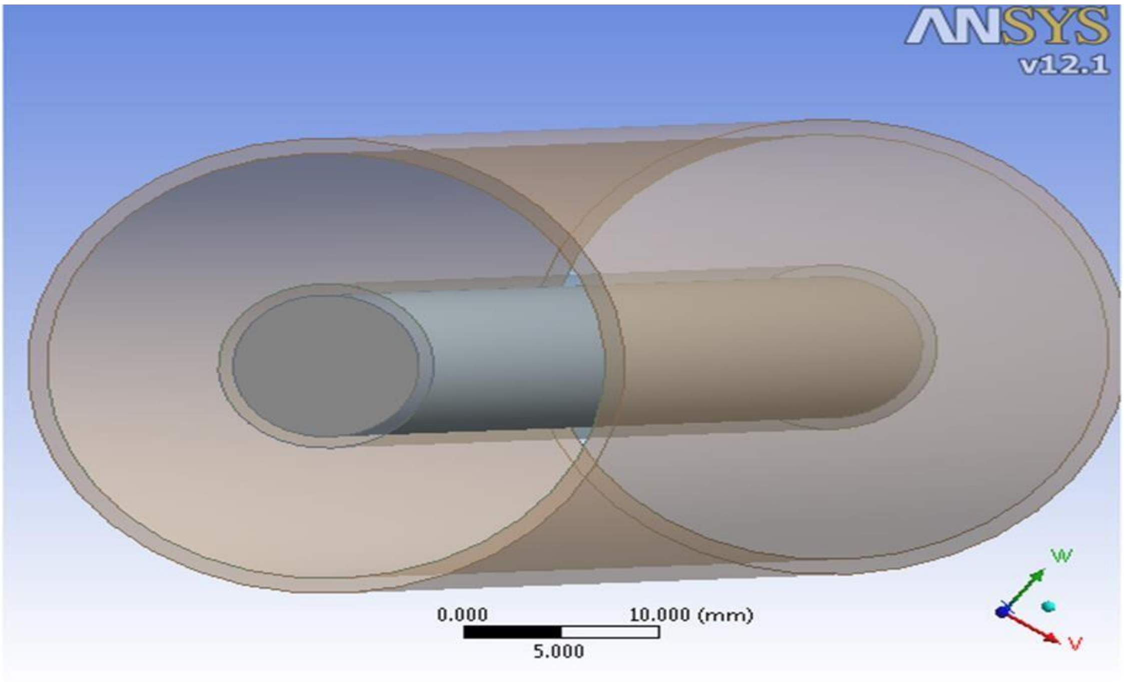


Figure 2:- Extrude Circle.

4.2.Mesh:-

Initially a relatively cover mesh is generated.This mesh contains mixed cells (Tetra and Hexahedral cells) having both trangular and quadrilateral faces at the bounderies. Care is taken to use structured Hexahedral cells as much as possible.It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner,particularly near well region. Later on, a fine mesh is generated. For this fine mesh, the edges and region of high temperature and pressure gradients are finely meshed.

4.2.1. y+ Values:-

y+ values plays a significant role in turbulence modelling for the near wall treatment. Y+ is a non dimensional distance.It is frequently used to describe how coarse or fine a mesh is for aparticular flow pattern. It determines the proper size of the cells near the domain walls. The turbulence model wall laws have limitations on the Y+ values at the wall. For instance, the standard K-epsilon model requires a wall y+ value between approximately 300 and 100. A faster flow near the wall must be decreased, y+ values for different wall treatments are given in table 2.

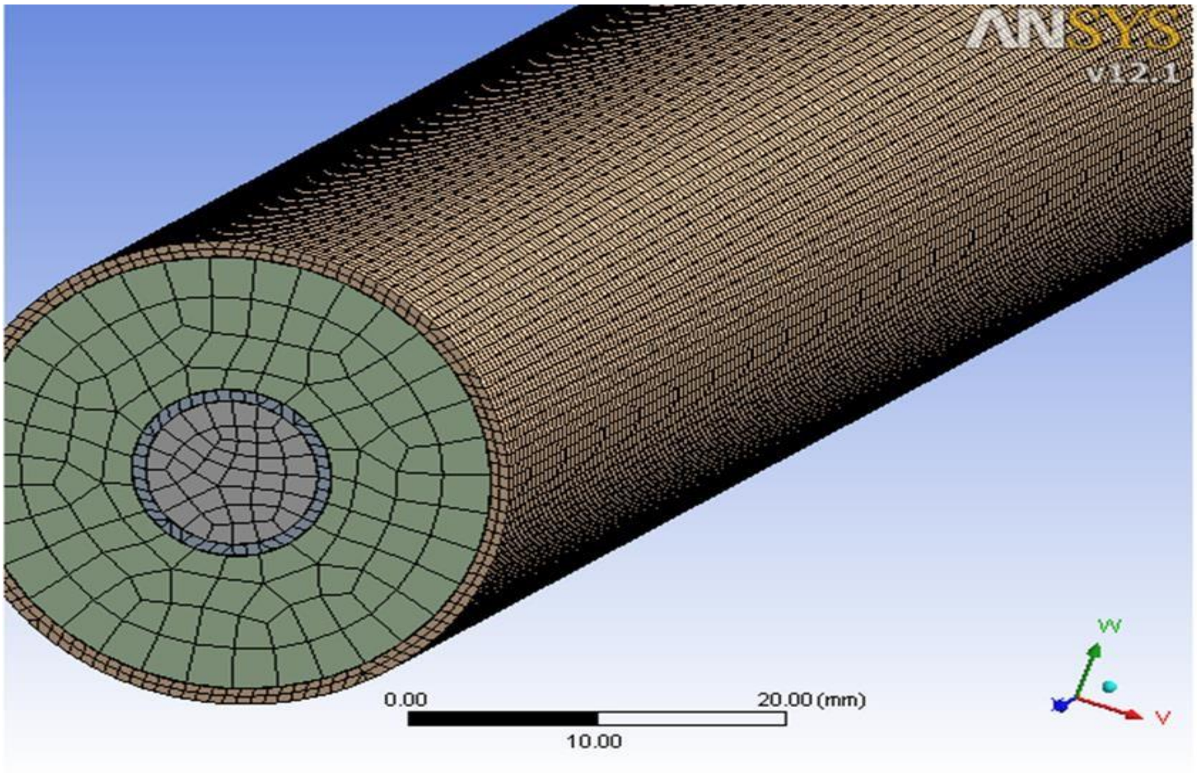


Figure 3: close view of mesh parts.

Table 2: y+ Values for the Different Wall Treatments:-

Wall treatment Method	Recommended Y+ Value	Used y+ Value at tube Wall
Standard Wall functions	30<y+<400	y+<5
Non equillibrium wall function	30<y+<400	y+<5
Low Reynolds no.model	y+=1	y+<1

The mesh Details gave us the following information:

- Relevance centre:fine meshing
- Smooting:high
- Size:0.747970mm to 74.7970mm
- Nodes: 1045694
- Elements: 703592

4.2.2. Named Selection:-

The different surfaces of the solid are named as per required inlets and outlets for inner and outer fluids. The outer wall is named as insulation surface. Save project again at this point and close the window. Refresh and update project on the workbench. Now open the setup. The ANSYS Fluent Launcher will open in a window. Set dimension as 3D, option as Double precision, processing as Serial type and hit OK. The Fluent window will open.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

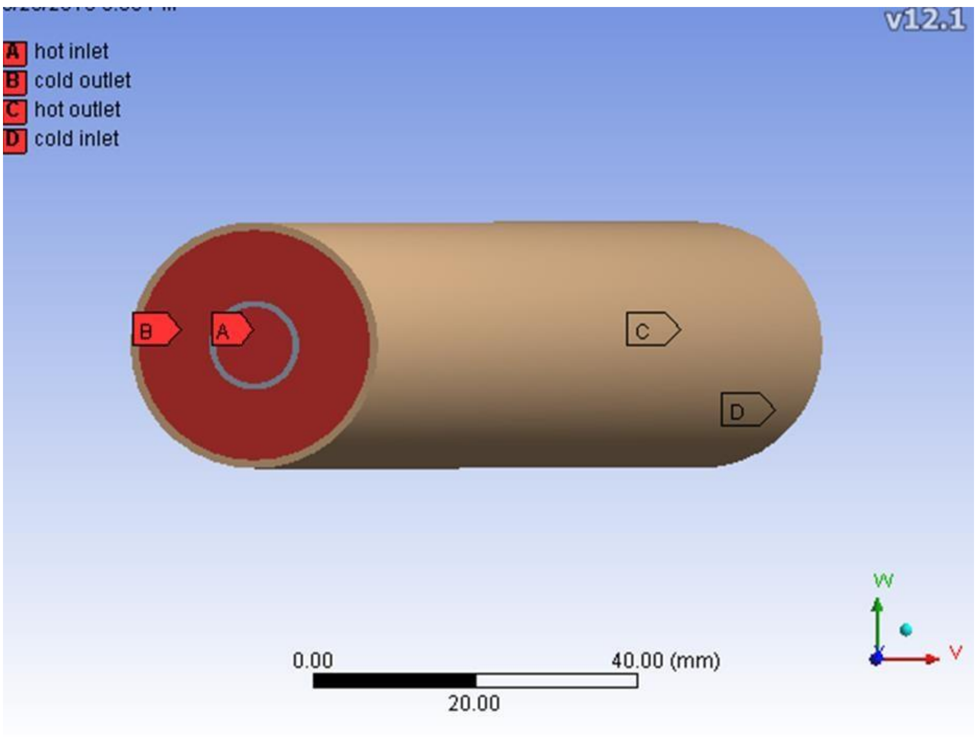


Figure 4: Named selection.

4.3. Solution:-

4.3.1. Problem Setup:-

The mesh is checked and quality is obtained. The analysis type is changed to pressure based type. The velocity formulation is changed to absolute and time to staedy state. Gravity is defined as y=(-9.81m/s^2).

4.3.2. Models:-

Energy is set to ON positions. Viscous model is selected as "k-Ɛ model".

4.3.3. Materials:-

The create/edit option is clicked to add water-liquid and copper to the list of fluid and solid respectively from the fluent database.

4.3.4. Cell zone conditions:-

Inner and outer fluid assigned as water and inner and outer solid pipe assigned as material copper.

4.3.5. Boundary Conditions:-

Boundary condition are used according to the need of the model. The inlet and outlet conditions are defined as mass flow inlet and pressure outlet. As this is a counter-flow and parrallel flow with two tubes as there are two inlets and two outlets. The walls are seperately specified with respective boundary conditions. No slip conditions is considered for each wall. Except the tube walls each wall is set to zero heat flux condition. The details about all boundary conditions can be seen in the table 3 as given below.

Table 3 Boundary Conditions for parallel and counter flow Hex

S.no.	Boundary Conditions Type	Mass Flow Rate Magnitude (kg/s)	Turbulent Intensity	Hydraulic Diameter	Temperature
Hot inlet	Mass flow inlet	0.02 kg/s	4.6%	**9.5mm**	**360k**
Hot outlet	Pressure outlet	-	4.6%	-	-
Cold inlet	Mass flow inlet	0.02 kg/s	3.6%	17.5mm	360k
Cold outlet	Pressure Outlet	-	3.6%	-	-

4.3.6. Reference Value:-

The inner inlet is selected from the drop down list of "compute from". The values are:

- Area=1m^2
- Density=998.2kg/m^3.
- Length=1000mm
- Temperature=360k
- Velocity=0.28m/s.
- Viscosity=0.001003kg/m-s.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

- Ratio of specific heats=1.4.

4.3.7. Solution Methods:-

The solution methods are specified as follows:-

- Scheme=Simplex.
- Gradient=Least Square Cell Based.
- Pressure=Standard
- Momentum=Power law
- Turbulent kinetic law=Power law.
- Turbulent Dissipation Rate=Power law
- Energy=Second order upwind.

4.3.8. Solution Control and Initialization:-

Under relaxation Factors the parametrs are:-

- Pressure=0.3 Pascal
- Density=1 kg/m^3
- Body forces=1 kg/m^2s^2
- Momentum=0.7kg-m/s.
- Turbulent kinetic energy=0.8m^2/s^2.

Then the solution initialization method is set to standard initialization whereas the reference frame is set to relative cell zone. The inner inlet is selected from the compute from drop down list and the solution is initialized.

4.3.9. Measure of Convergence:-

It is tried to have a nice convergence throughout the simulation and hence criteria is made strict so as to get an acuurate result.For this reason residuals are given as per the table 4 that follows.

Variable	Residual
X-Velocity	10^{-6}
Y-Velocity	10^{-6}
Z-Velocity	10^{-6}
Continuity	10^{-6}
Specific dissipation energy/Dissipation energy.	10^{-5}
Turbulent Kinetic Energy	10^{-5}
Energy	10^{-9}

4.3.10. Run Calculation:-

The number of iteration is set to 300 and the solution is calculated and various contours, vectors, and place are obtained.

5.RESULTS AND DISCUSSION:-

5.1.Contours:-

The temperature,pressure and velocity distribution along the heat exchanger can be seen through the contours.

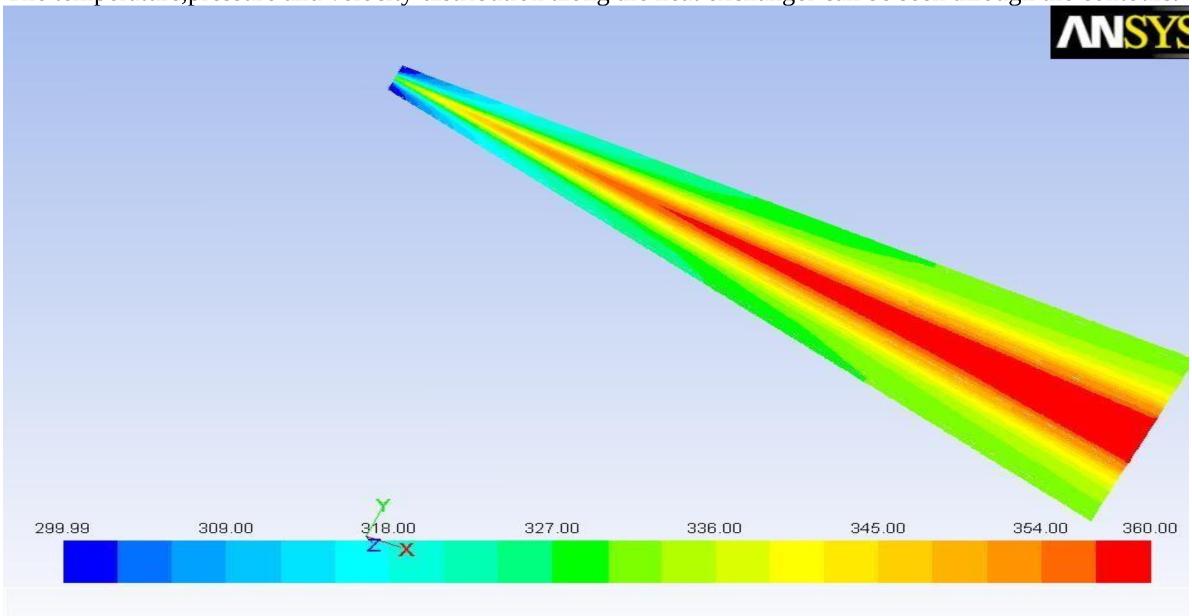


Figure 5: contours of static temperature in counter flow hex in Kelvin.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

Mass Flow Rate.	For Counter hex (in kg/sec)
Hot Inlet	0.02
Cold Inlet	0.02
Hot Outlet	-0.00099999554
Cold Outlet	-0.0010000009

Table 5: mass flow rate.

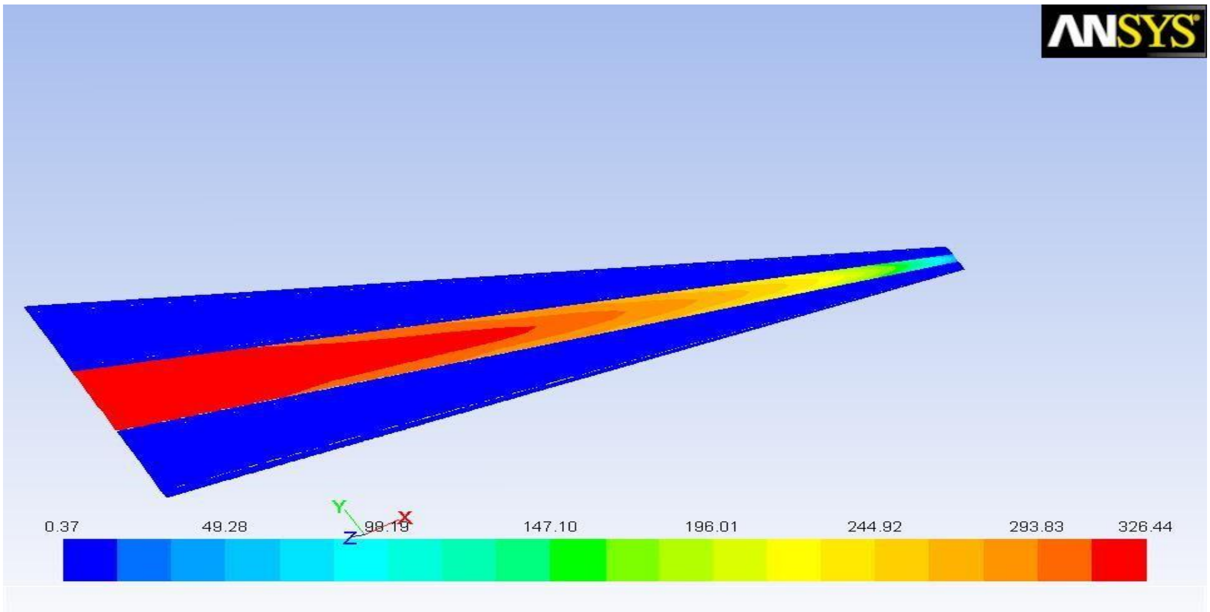
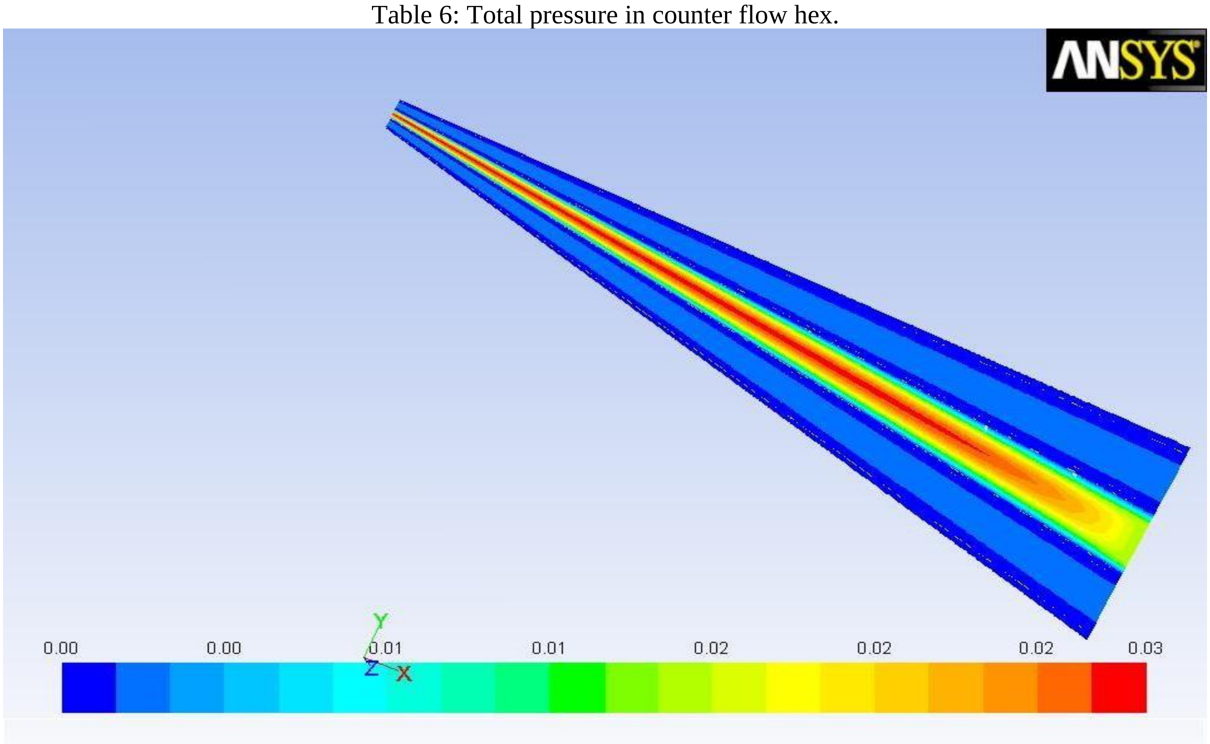


Figure 6: Contours of total pressure for Counter Flow hex(pascal)

Pressure(Pascal)	For Counter hex (Pascal)
Hot Inlet	7.47
Cold Inlet	0.3811
Hot Outlet	0.13
Cold Outlet	0.0021

Table 6: Total pressure in counter flow hex.



5.2. X-Y plots:-

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

XY-PLOTS OF TEMPERATURE:

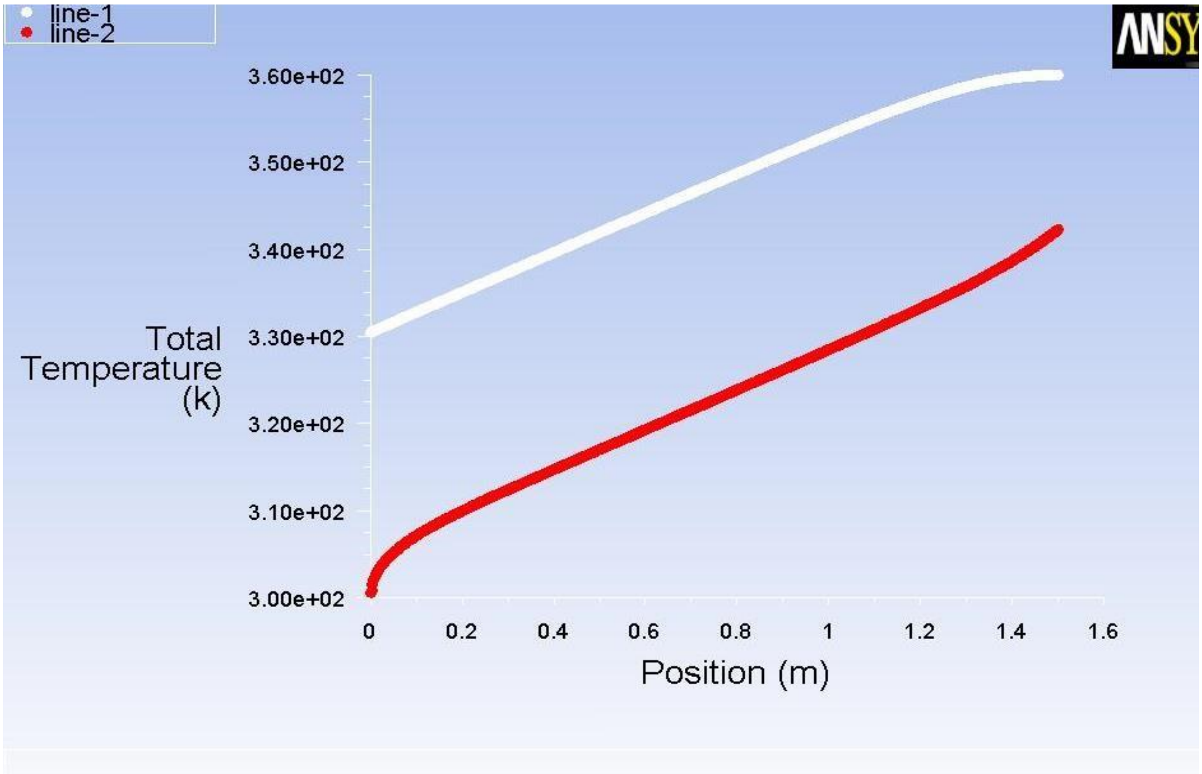


Figure 8: Plot of static temperature of counter flow.

XY-PLOT OF TOTAL PRESSURE:

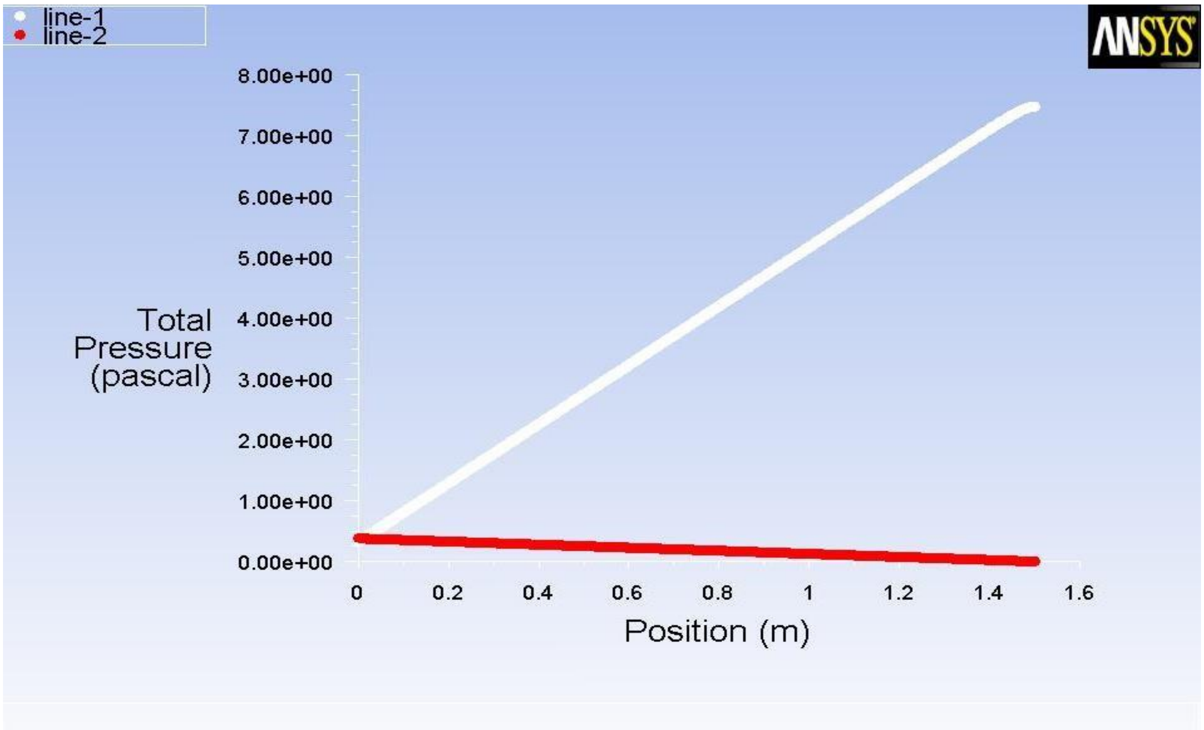


Figure 9: Plot of total pressure of counter flow hex.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

XY-PLOT OF VELOCITY:

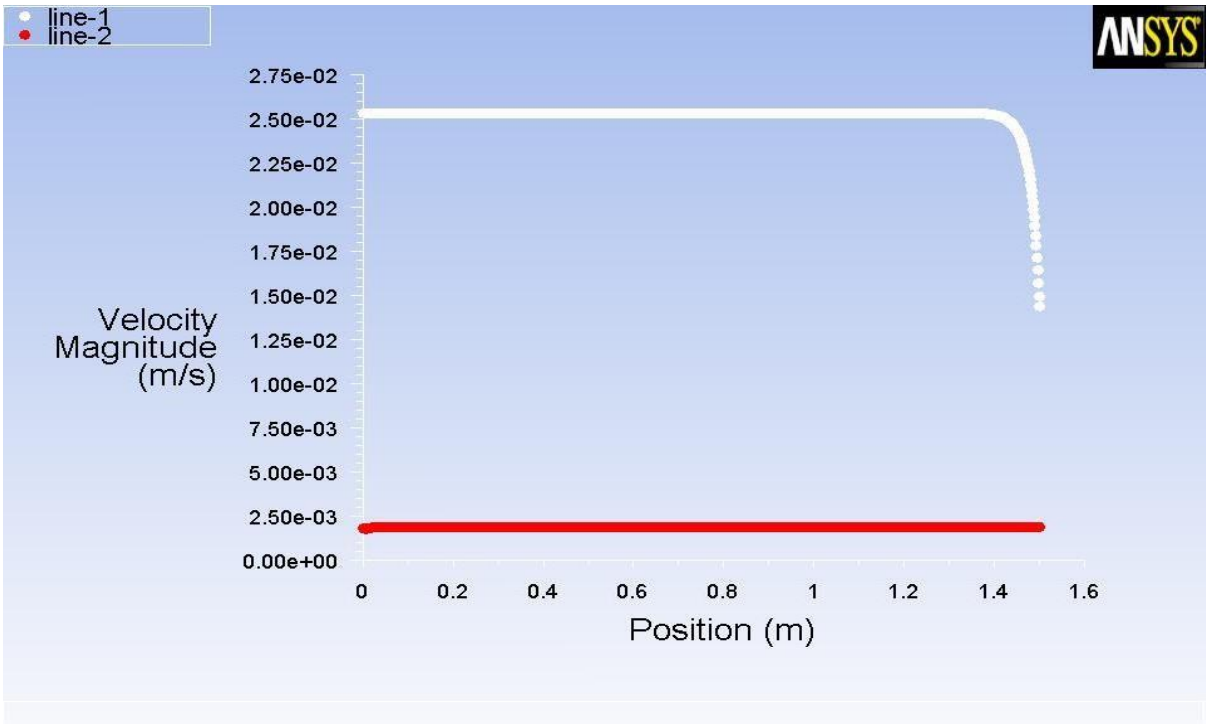


Figure 10: Plot of Velocity magnitude For Counter Flow (m/s)

6.REPORT:-

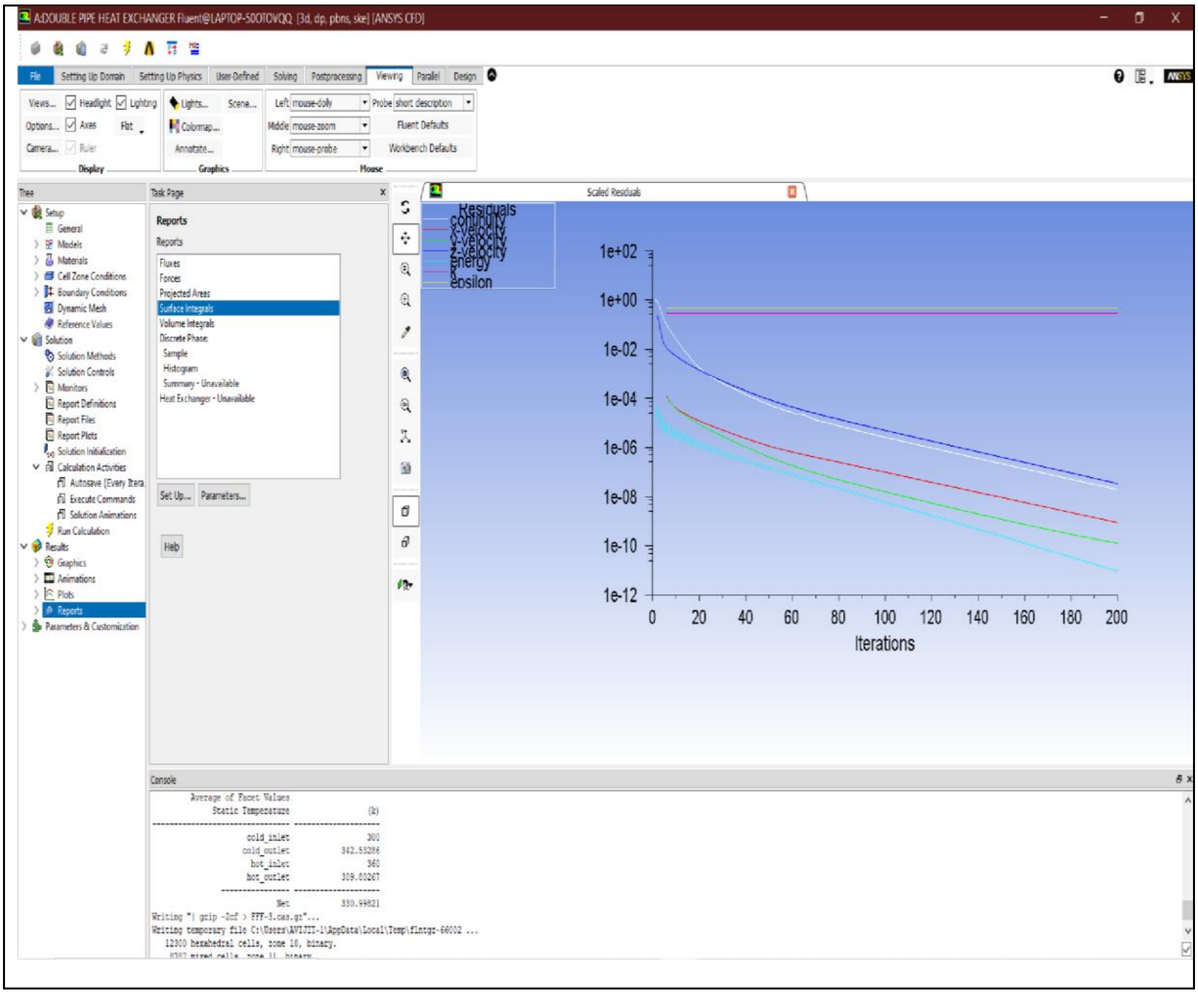


Figure 11: variation of velocity, temperature, energy etc.

POSITION	INLET TEMPERATURE (K)	OUTLET TEMPERATURE (K)
COLD	300	342.53
HOT	300	309.80

Table 7: Temperature at inlet and outlet.

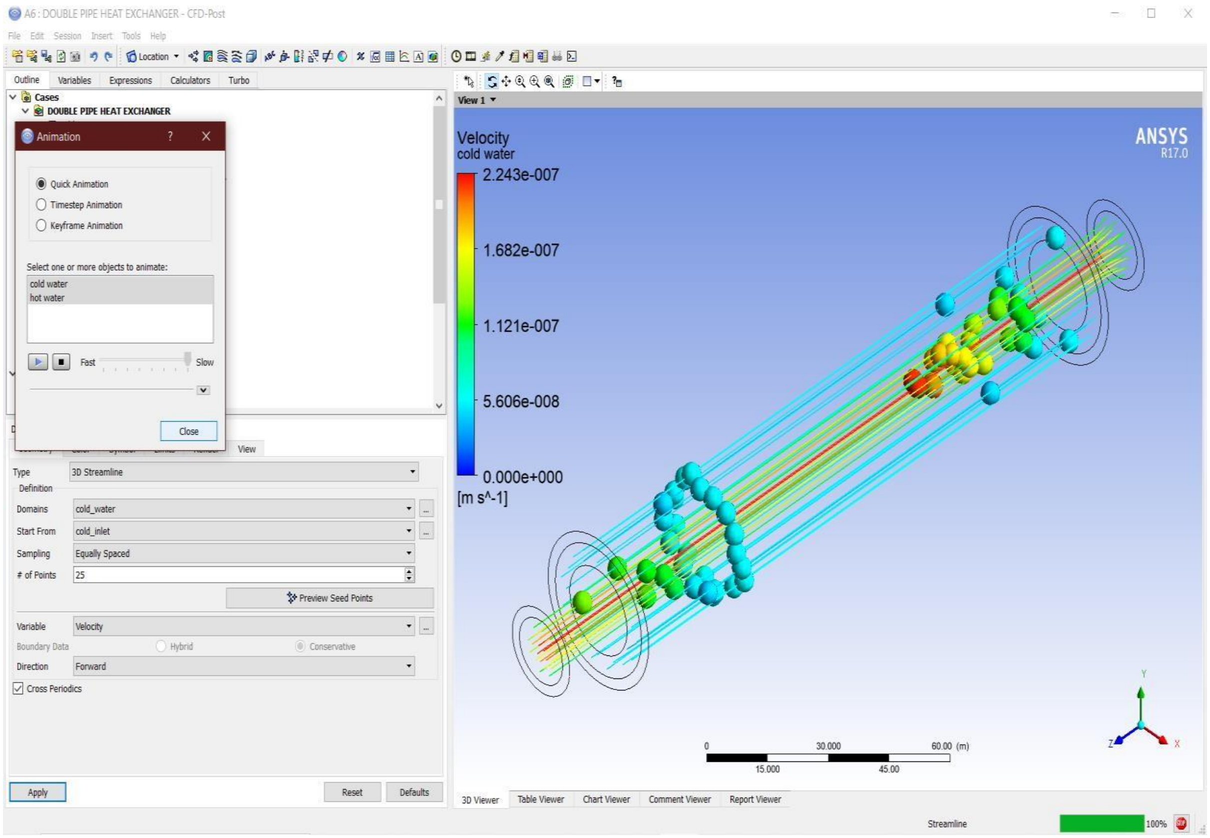


Figure 12: flow animation

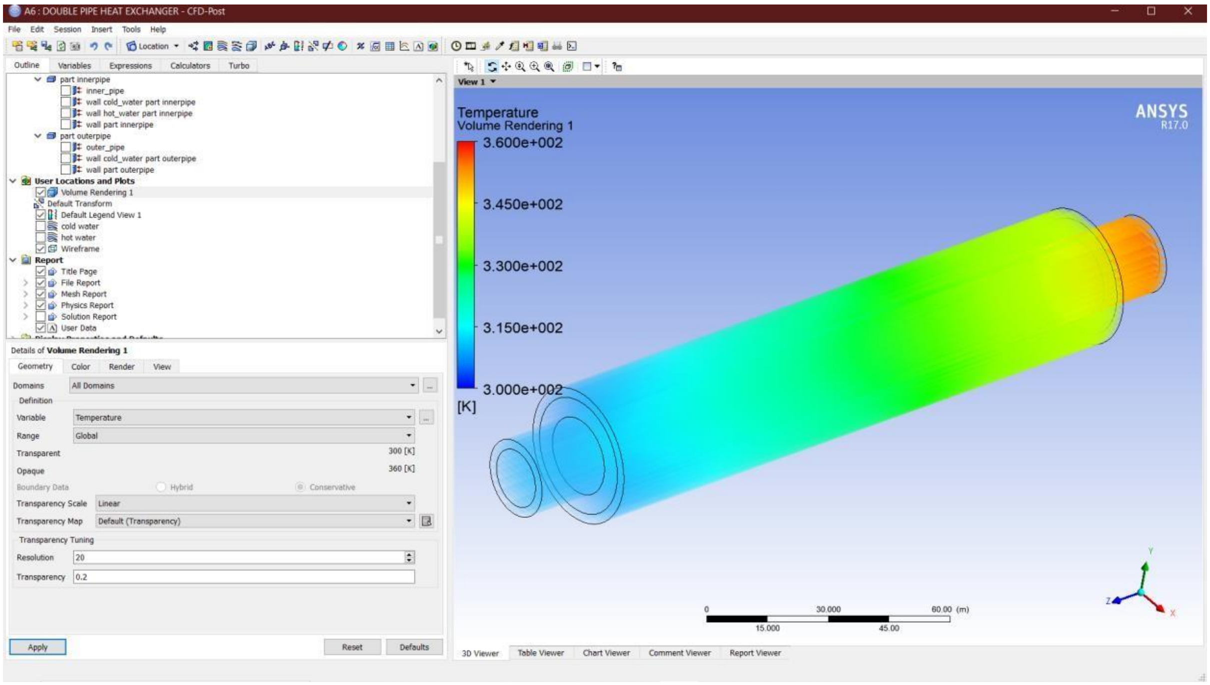


Figure 13: Temperature distribution.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

7.CONCLUSION:-

A CFD package (ANSYS FLUENT 17.0) was used for the numerical study of heat transfer characteristics of a double pipe heat exchanger for counter flow, and the results was detrmined. The study showed the heat transfer within the error limits performance of counter-flow configuration. Nusselt number at different points along the pipe length was determined from the numerical data. The simulation was caried out for water to water heat transfer characteristics and for same length and same diameter of tube and annulus for same input temperature for cold inlet 300K, and for hot inlet 360K. We analyse that in counter flow heat exchanger there is high temperature difference in output streams (hot outlet, cold outlet). Nusselt number for the counter flow heat exchanger 802.

For the given design and length of heat excahnger heat transfer enhancement in a double pipe heat exchanger is possible achieved by several methods.These techniques are divided into active and passive techniques.Active methods involve some external input for the enhancement of the heat transfer like induced vibrations, injections and suctions of the fluids and jet impingement etc. Another method is the passive method without the simulation by external power such as surface coating, surface roughness and extended surfaces.

8.REFERENCES:-

[1] Heat Transfer in Process Engineering by EDUARDO CAO.
[2] Heat Exchanger Selection, Rating, and Thermal design (Second Edition), by Sadik Kakac
[3] Computational Fluid Dyanmics with the applications, by John D.Anderson.
[4] Fluid Mechanics,by S.K.Som and Biswas.
[5] White Frank, Viscous Fluid Flow, 3rd Edition New york: McGraw Hill Companies, Inc 2006.
[6] Kays, William, Micheal Crawford, and Bernhand Weigand. Convective heat and Mass Transfer 4th Edition New York: McGraw hill Companies, Inc 2005.
[7] Numerical study of heat transfer in a finned double pipe Heat exchanger, Shiva Kumar, K. Vasudev Karanth, Krishna Murty Dept of Mechanical Engg, MIT, Manipal India (October 13,2014).
[8] A Review on Design & Analysis of Double pipe heat exchanger N.R.Choudhuri,F.N. Adroja Mechanical Engineering Department,RK University Rajkot, Gujurat India.
[9] Heat Transfer by Y.A.Cengel.
[10] ANSYS FLUENT manual.