International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 06 | June - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

CFD Analysis, Of Double Pipe Parallel Flow Heat Exchanger

Mrs. K. Deepika[1], D. Saiprasad[2], E. Surya Prakash Goud[3], G. Pranay Kumar[4]

[1]Department of mechanical engineering & Guru Nanak Institute of Technology
[2]Department of mechanical engineering & Guru Nanak Institute of Technology
[3]Department of mechanical engineering & Guru Nanak Institute of Technology
[4]Department of mechanical engineering & Guru Nanak Institute of Technology

---***---

Abstract - Heat exchanger is a device that exchange heat between two fluids of different temperature that can have solid barrier or directly mix the two fluids. The heat exchange process happens by one of the three principles such as conduction, convention, and radiation. Here radiation does not play a major role. Heat ex-changer used in many industries to transfer heat between hot and cold fluid. Here in this project, we will design and analyze counter flow of fluid in double pipe heat exchanger, basically in this project we are trying to find the best performance of double pipe heat exchanger in terms of the heat transfer rate by varying some of its parameter such as flow rate of the fluid in the heat exchange and varying the diameter of the pipe and length of the pipe. In this particular design we have used the inside pipe made of aluminum, the outside or annular pipe made of aluminum, the hot fluid and the cold fluid we have taken is water. We are using ANSYS fluid fluent analysis software for both designing the double pipe heat exchanger and to analyze the counter flow performance of the double pipe heat exchanger.

***Key Words*:** Heat Exchanger, Convection, Conduction, Radiation, ANSYS, CFD, Module

1.INTRODUCTION

A heat exchanger is a device used to transfer heat between a solid object and a fluid, or between two or more fluids. The fluids may be separated by a solid wall to prevent mixing, or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning, power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment. The classic example of a heat exchanger is found in an internal combustion engine in which a circulating fluid known as engine coolant flows through radiator coils and air flows past the coils, which cools the coolant and heats the incoming air.

2.LITERATURE SURVEY

Ahmerrais khan and Sarfaraz khan focus on the various researches on Computational Fluid Dynamics (CFD) analysis in the field of heat exchanger. It has been found that CFD has been employed for the various areas of study in various types of heat exchanges Different turbulence models available in general purpose commercial CFD tools i.e., Standard, realizable and RNG k −ε RSM, and SST k −ε in conjunction with velocity-pressure coupling schemes such as SIMPLE, SIMPLEC, PISO etc.Have been adopted to carry out the simulations. The quality of the solutions obtained from these simulations are largely within the acceptable range proving that CFD is an effective tool for predicting the behaviour and performance of a wide variety of heat exchangers.

• **Philippe Wildi**-Tremblay in his paper explains the procedure for minimizing the cost of a shell-and-tube heat exchanger based on genetic algorithms (GA). The global cost includes the operating cost (pumping power) and the initial cost expressed in terms of annuities. He took some geometrical parameters of the shell-and-tube heat exchanger as the design variables and the genetic algorithm is applied to solve the associated optimization problem. It is shown that for the case that the heat duty is given, not only can the optimization design increase the heat exchanger effectiveness significantly, but also decrease the pumping power dramatically.

• **Pignotti** in his paper established relationship between the effectiveness of two heat exchanger configurations which differ from each other in the inversion of either one of two fluids. This paper provides the way by which if the effectiveness of one combination is known in terms of heat capacity rate ratio and ntus then the effectiveness of the other combination can be readily known.

3.PROBLEM STATEMENT AND METHODOLOGY

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

3.2 Methodology:
Problem statement
Selection of software
Modelling of double pipe heat exchanger
Analysis using CFD

Results and discussion

4.SOFTWARES-SOLIDWORKS

Solid Works is a 3D solid modelling package which allows users to develop full solid models in a simulated environment for both design and analysis. In Solid Works, you sketch ideas and experiment with different designs to create 3D models. Solid Works is used by students, designers, engineers, and other professionals to produce simple and complex parts, assemblies, and drawings. Designing in a modelling package such as Solid Works is beneficial because it saves time, effort, and money that would otherwise be spent prototyping the design.

4.1 Solid Works Components - Parts

Before we begin looking at the software, it is important to understand the different components that make up a Solid Works model.

Part:

➢ The first and most basic element of a Solid Works model is a Part.

➢ Parts consist of primitive geometry and feature such as extrudes, revolutions, lofts, sweeps, etc.

➢ Parts will be the building blocks for all the models that you will create Assembly:

➢ The second component is the assembly. Assemblies are collections of parts which are assembled in a particular fashion using mates (constraints).

➢ Any complex model will usually consist of one, or many assemblies. Drawing:

➢ The third and final component in Solid Works is the Drawing.

➢ A drawing is the typical way to represent a 3D model such that any engineer (or manufacturer) can recreate your part. Drawings are important because they provide a standard way of sharing your design

4.2 Solid Works – Let's Begin

➢ By default, no file is opened automatically when you start the program.

➢ New or click the New File icon in the main toolbar.

➢ This will open the New Solid Works document wizard.

➢ Let's begin by creating a new part.

➢ To do this, click on Part, then OK. Once you do this, you will be brought into the modeling view which should open several toolbars and panes.

4.3. Terminology These terms appear throughout the Solid Works software and documentation.

➢ Origin: Appears as two blue arrows and represents the (0, 0, 0) coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the (0, 0, 0) coordinate of the sketch. You can add dimensions and relations to a model origin, but not to a sketch origin.

➢ Plane: Flat construction geometry. You can use planes for adding a 2D sketch, section view of a model, or a neutral plane in a draft feature, for example.

➢ Axis: Straight line used to create model geometry, features, or patterns. You can create an axis in different ways, including intersecting two planes. The Solid Works application creates temporary axes implicitly for every conical or cylindrical face in a model.

➢ Face: Boundaries that help define the shape of a model or a surface. A face is a selectable area (planar or non-planar) of a model or surface. For example, a rectangular solid has six faces.

➢ Edge: Location where two or more faces intersect and are joined together. You can select edges for sketching and dimensioning.

➢ Vertex: Point at which two or more lines or edges intersect. You can select vertices for sketching and dimensioning.

4.4. User Interface

The Solid Works application includes user interface tools and capabilities to help you create and edit models efficiently, including Windows Functions: The Solid Works application includes familiar Windows functions, such as dragging and resizing windows. Many of the same icons, such as print, open, save, cut, and paste are also part of the Solid Works application.

4.5 Solid Works Document Windows:

Solid Works document windows have two panels. The left panel, or Manager Pane, contains:

4.6 Feature Manager /Design Tree Specification Tree 11 Displays the structure of the part, assembly, or drawing. Select an item from the Feature Manager Design tree to edit the underlying sketch, edit the feature, and suppress and un suppress the feature or component, for example. Property Manager: Provides settings for many functions such as sketches, fillet features, and assembly mates. Figure6. Extrude Toolbox Configuration Manager Lets you create, select, and view multiple configurations of parts and sub document. assembly within a single document. For example, you can use configurations of a bolt to specify different lengths and diameters. part or assembly within a single document. For example, you can use configurations of a bolt to specify different lengths and diameters.

4.7 Mouse Buttons. Configuration Box Mouse buttons operate in the following ways:

➢ Left: Selects menu items, entities in the graphics area, and objects in the Feature Manager Design tree.

➢ Right: Displays the context-sensitive shortcut menus.

➢ Middle: Rotates, pans, and zooms a part or an assembly, and pans in a drawing.

➢ Mouse gestures: You can use a mouse gesture as a shortcut to execute a command, similar to a keyboard shortcut. Once you learn command mappings, you can use mouse gestures to invoke mapped commands quickly. To activate a mouse gesture, from the graphics area, right-drag in the gesture direction that corresponds to the command. When you right-drag, a guide appears, showing the command mappings for the gesture directions. An assembly is a combination of two or more parts, also called components, within one SOLIDWORKS document. You position and orient components using mates that form relations between components. This lesson discusses the following:

➢ Adding parts to an assembly.

➢ Moving and rotating components in an assembly.

➢ Creating display states in a product of ANSYS inc. Is a world's leading, widely distributed, and popular commercial CAE package. It is widely used by designers/analysis in industries such as aerospace, automotive, manufacturing, nuclear, electronics, biomedical, and much more. ANSYS provides simulation solution that enables designers to simulate design performance directly on the desktop. In this way, it provides fast efficient and cost-efficient product development from design concept stage to performance validation stage of product development cycle. It helps to acceleration and streamlines the product development process by helping designers to resolve issues relation to structural thermal fluid flow electromagnetic effect a combination of these phenomena acting together and soon. In ANSYS, the basics of FEA concepts, modelling and the analyzing of engineering problem using ANSYS workbench. In addition, describe of importance tools and concepts given whenever required. this following simulation streams of ANSYS.

1. Structural Analysis Static Structural Analysis Modal Analysis Transient Structural Analysis
2. Thermal Analysis Steady State Thermal Analysis Transient

4.8 Thermal Analysis

Project Objectives After completing of this chapter,

- ➢ The basic concept and general working of FEA.
- ➢ Understand the advantage and limitations of FEA.
- ➢ Understanding the analysis type.
- ➢ Understanding important terms and definition of FEA

4.9 Introduction Of FEA

The finite element analysis (FEA) is computing technique that is used to obtain approximately. Solution to boundary valve problems. It uses a numerical method called finite element method(fem).in FEA involves the computer model of a design that is loaded and analyzed for specific results, much as stress, deformation, deflection, natural frequencies, mode shapes, temperatures distribution and soon. The concept of FEA can be explained through a basic measurement of dimensions. In FEA simulation the loading condition of design and determination the design responses in those conditions. It can be used in new product design as well as in exiting product refinement. A model is divided into a finite number of regions/ divisions call elements, these elements can be of predefinition shapes, such as triangular, quadrilateral, hexahedron, tetrahedron and soon. This predefined shape(elements) helps to find the sum of responses of all elements in model gives the total responses of complete model .General Working of Fea A better knowledge of FEA helps in building the most accurate model. Also, it helps in understanding the backwards working of ANSYS. Here, a simple model is discussed to give a brief overview of working of FEA. A spring assembly that represents a simple two spring element model. In this model, two spring are connecting in series and one of the springs is fixed at the leftmost endpoint. The stiffness of the spring constant k1 and k2.

The movement of the endpoint of each spring is restricted to x direction only. The change of position from the undeformed state of each endpoint can be definition by variables x1 and x2. The forces acting on the end point of spring are represented by f1 and f2. Representation of A Two Spring Assembly. To develop a model that can predict the state of this spring assembly, you can use the linear spring equation given below F=KX Where, F - Force, K- Spring Constant and X - displacement If we use the spring parameters defined above and assume a state of equilibrium, the following equation can be written for the state of each endpoint.

F1= (K1+K2) X1+(-K2) X2

F2=(-K2) X1+(K2)X2

Therefore, [$f1$ $k1 + k2$ $-k2$ $x1$] =[$f2$ $-k2$ $k2$]

In above matrix, if the spring constant is k1and k2 are known and the deformation shapes x1 and x2 are defined. Then the resulting forces f1 and f2 can be determined. Similarly for all terms.

4.10 Stiffness Matrix

In above equation, the following part represent the stiffness matrix(k) $k1 + k2 \cdots -k2$ [$\vdots \ddots \vdots$] $-k2 \cdots k2$ This matrix is relatively simple because it comprises only one pair of springs, but it turns complex when the number of spring increase.

4.11 Degree Of Freedom The degree of freedom is defined as the least number of independent coordinates required to define the configuration of the system in space. In the previous example, we are only concerned with displacement and forces. By making one endpoint fixed, you will restrict all degrees of freedom for that node. Which means that, there will be no translational or rotational degrees of freedom for that node.

4.12 Boundary Conditions

The boundary conditions are used to elimination the unknowns in the system. At equations that are solvable is meaningless without the input. So, the displacement could have been specified in place of forces as boundary conditions and the mathematical model could have been solved for the forces. In other words, the boundary condition can help to reduce or eliminate. unknown in the boundary.

4.13 Elements And Elements Shapes

Elements is an entity into which the system under study is divided. An element shape is specified by nodes. The shape (area, length, and volume) of elements depends on the nodes with which it is made. An element (triangular shape) is shown in the figure. A Node and An Element Element shape There are many types of elements shapes that are further divided into various classes depending on their uses. The following are some basic elements shape Line element line element has a shape of line or curve. Therefore, a minimum of two nodes is required to define it. The higher order elements those additional nodes at the middle of the line or curve. An element the does not have a node in between its edges are called linear element Area elements area elements have a shape of triangular or a quadrilateral, therefore it's required a minimum of three or four nodes to define it. The Area Elements Volume elements volume elements have the shape of a hexahedron, wedge, tetrahedron, or a pyramid. olume Elements .

4.14 General Procedure to Conduct Finite Elements Analysis

To conduct the finite elements analysis, you need to follow certain steps that are given next.

1. Set the type of analysis to used.
2. Create model
3. Define the elements type
4. Divide the element types
5. Divide the given geometry into nodes and elements
6. Apply material properties and boundary conditions 7. Drive elements matrices and equations
8. Solve the unknown parameters at nodes
9. Interpret the results

4.15 Pre-processor

The pre-processor is a phase that process input data to produce an output which is used as input in the subsequent phase (solution). Following are the input data that need to be given to the pre-processor.

➢ Real constants for elements (cross-sectional area, moments of inertia, shell thickness and soon).

➢ Material properties (young's modulus, Poisson's ratio, spring constant, thermal conductivity, coefficient of thermal expansion, and soon).

➢ Geometry model (either created in FEA software or imported from cad package).

➢ FEA model (discretizing the geometric model into small elements).

➢ Loading and boundary conditions (defining loads, pressures, moments, temperature, conductivity, convection, constraints (fixed, pinned, or frictionless/ symmetrical)), and so on. The input data are pre-processed for the output data and the pre-processor generates the data files automatically with help of uses.

Solution The solution phase is completely automatic. The FEA software generates elements matrices, compute nodal value and derivatives, and stores the result data in files. These files are further used in the subsequent phase (postprocessor) to review and analyses the results through the graphics display and tabular listing.

4.16 Postprocessor

The output from the solution phase (results in data files) is in the numerical form and consist of nodal values of the field variable and its derivative. For example, in structural analysis, the output of the post processor is nodal displacements and stress in elements. The postprocessor processes the results data and displays them in graphical form to check or analysis the results. The graphical output gives the detailed information about the required results data. The postprocessor phase is automatic and generates graphical output in the specified form.

4.16 Fea Software

There is a variety of commercial FEA software package available in the market. Every CAE software provides various modules for various analysis requirements. Depending on your requirement, you can select a required module for your analysis. Some firms use one or more 17 CAE software and other develop customized version of the commercial software to meet their requirement.

4.17 Advantage And Limitations of Fea Software Following is some of the advantage and laminations of FEA software.

4.18 Advantages

It reduces the amount of prototype testing, thereby saving the cost and time. • It gives the graphical representation of the results of analysis • The finite elements modelling and analysis are performed in the preprocessor and solution phases which if done manually would consume a lot of time and in some cases, might be impossible to perform. • Variables such as stress and temperature can be measured at any desired point of the model.

• It helps optimise a design.

• It is used to simulate the designs that are not suitable for prototype testing.

• It helps you create more reliable, high quality, and competitive designs.

4.19 Limitations

It does not provide exact solutions

• FEA packages are costly

• An inexperienced user can deliver incorrect answer, upon which expensive decision will be based

• Results give solutions but not remedies such as bolts, welded joints, and so on cannot be accommodated to model. This may lead to approximation and errors in the results.

• For more accurate results, more hard disk space, ram, and time are required

4.20 Types Of Engineering Analysis

You can perform different types of analysis using FEA software and these are discussed next. Structural Analysis In structural analysis first the nodal degrees of freedom (displacements) are calculated and then the stress, strains, and reaction forces are calculated from nodal displacements. The classification of structural analysis is shown. Static Analysis: In static analysis, the load or field conditions do not vary with respect to time, and therefore, it is assumed that the load or field conditions are applied gradually, not suddenly. The system under this analysis can be linear or non-linear. The inertia and damping effects are ignored in the structural analysis. In structural analysis, the following matrices are solved. $[K][X] = [F]$

Where, F - Force, K- Spring Constant and X -Displacement The above equations are called the forced balanced equation for the linear system. If the elements of matrix [k] are the function of [x], the system is known as the non-linear system.

The outputs that can be expected from FEA software are given

• Displacements (deformation)

• Stress

• Strains

• Reaction forces

4.21 Thermal Analysis

The thermal analysis is used to determine the temperature distribution and related thermal quantities such as thermal distribution, the amount of heat loss or gain, thermal gradients, and thermal fluxes. All primary heat transfer modes such as conduction, convection, and radiation can be simulated. You can perform two types of types of thermal analysis, steady state and transient. Steady State Thermal Analysis In this analysis, the system is studied under steady thermal loads with respect to time. Transient Thermal Analysis In this analysis, the system is studied under varying thermal loads with respect to time. Fluid Flow Analysis This analysis is used to determine the flow distribution temperature of fluid. The ANSYS/FLOWTRAN program is used to simulate the laminar and turbulent flow, compressible and electronics packaging, automotive design, and soon. The output that can be expected from the fluid flow analysis are velocities, pressures, temperatures, and film coefficients.

4.22 Important Terms and Definitions

Some of die important terms and definition used in FEA are discussed next. Strength When a material is subjected to an external load, the system undergoes a deformation. The material, in turn, offers resistance against this deformation. This resistance is offered by virtue of the strength of the material. Load: The external pressure acting on a body is called load. Stress: The pressure of resistance offered by a body against the deformation is called stress. The stress is induced in

the body while the load is being applied on the body. The stress 1S Calculated 85 load per unit area.
P = F/A
Where, P = Stress in N/mm’
F = Applied Pressure in Newton
A = Cross-Sectional Area in mm’
• The material can undergo various types of stresses, which are discussed next. Tensile Stress: If the resistance offered by a body is against the increase in the length, the body is said to be under tensile stress. Compressive Stress: If the resistance offered by a body is against the decrease in the length, the body is said to be under compressive stress.
• Compressive stress is just the reverse of tensile stress. Shear Stress: The shear stress exists when two materials tend to slide across each other in any typical plane of shear on the application of pressure parallel to that plane. 19 Shear Stress = Shear resistance (R) / Shear area (A) Strain: When a body is subjected to a load (force), its length changes. The ratio of change in the length of the body to its original length is called strain. If the body returns to its original shape on removing the load, the strain is called elastic strain. If the body remains distorted after removing the load, the strain is called plastic strain. The strain can be of three types, tensile, compressive, and shear strain. Strain (e) = Change in Length (dl) / Original Length (l) Elastic Limit: The maximum stress that can be applied to a material without producing the permanent deformation is known as the elastic limit of the material. If the stress is within the elastic limit, the material returns to its original shape and dimension on removing the external stress. Hooke’s Law: -It states that the stress is directly proportional to the strain within the elastic limit. Stress/ Strain = Constant (within the elastic limit) Young’s Modulus or Modulus of Elasticity: -In case of axial loading, the ratio of intensity of the tensile or compressive stress to the corresponding strain is constant. This ratio is called Young's modulus, and is denoted by E. E = p/e Shear Modulus or Modulus of Rigidity: In case of shear loading, the ratio of shear stress to the corresponding shear strain is constant. This ratio is called Shear modulus, and it is denoted by C, N, or G. Ultimate Strength: The maximum stress that a material withstands when subjected to an applied load is called its ultimate strength. Factor of Safety: The ratio of the ultimate strength to the estimated maximum stress in ordinary use (design stress) is known as factory of safety. It is necessary that the design stress is well below the elastic limit, and to achieve this condition, the ultimate stress should be divided by factory of safety. Lateral strain: If a cylindrical rod is subjected to an axial tensile load, the length (l) of die rod will increase (dl) and the diameter (dia) of the rod will decrease (do). In short, the longitudinal stress will not only produce a strain in its own direction but will also produce a lateral strain. The ratio dl/l is called the longitudinal strain or the linear strain, and the ratio d0/ (0) is lateral strain. Poisson's Ratio: The ratio of the lateral strain to the longitudinal strain is constant within the elastic limit. This ratio is called the Poisson’s ratio and is denoted by 1/m. For most of the metals, the value of the ‘m’ lies between 3 and 4. Poisson’s ratio = Lateral Strain/ Longitudinal Strain = 1/m Bulk Modulus: If a body is subjected to equal stresses along the three mutually perpendicular directions, the ratio of the direct stresses to the corresponding volumetric strain is found to be constant for a given material when the deformation is within a certain limit. These 20 ratios are called the bulk modulus and is denoted by K. Stress Concentration The value of stress changes abruptly in the regions where the cross-section or profile of a structural member changes abruptly. The phenomenon of this abrupt change in stress is known as stress concentration and the region of the structural member that is affected by stress concentration is known as the region of stress concentration. The region of stress concentration needs to be meshed densely to get accurate results. Bending: When a non-axial pressure is applied on a structural member; the structural member starts deforming. This phenomenon is known as bending. In case of bending, strains vary linearly from the centerline of a beam to the circumference. In case of pure bending, the value of strain is zero at the centerline. The plane section of the beam is assumed to remain plain even after the bending Bending Stress: When a non-axial pressure is applied on a structural member, some compressive and tensile stresses are developed in the member. These stresses are known as bending stresses. Creep: At elevated temperature and constant load, many materials continue to deform

Table -4.1: Analysis

Name of analysis	Application of loads	Solution determines
Explicit dynamics	Loads with respect to time	Total deformation or impact deformation
Fluid flow (CFX)	Compressible or incompressible of air or gases	Heat transfer or flow of air
Harmonic response	Periodic or sinusoidal loads	Resonance, fatigue, and effect of forced vibratio
Rigid dynamic	Constraints and motion load	Forces or direction analysis
Static structura	Static load conditions	Deformation, Stresses and Strains, Fatigue tool, Life, Damages, Safety factor
Steady state thermal	Heat flux or temperatures	Heat flux or temperatures
Transient structural	Heat flux or temperatures	Deformation, Stresses and Strains, Fatigue tool, Life, Damages, Safety factor
Transient thermal	Varying of Temperature or thermal loads with changing of times	Deformation status
Heat flux	stain	Heat

Table.5.2 temperature values

temperature	For parallel hex	For counter hex
Cold inlet	300	300
Hot inlet	360	360
Hot outlet	348	321
Cold outlet	312	336

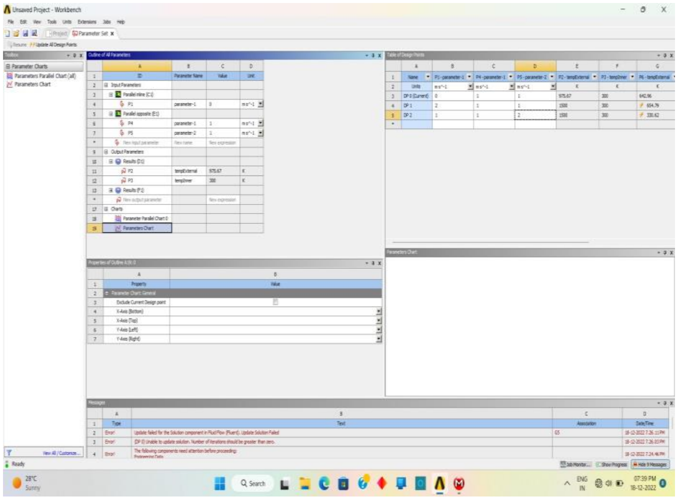

Figure 5.1 Analysis of design

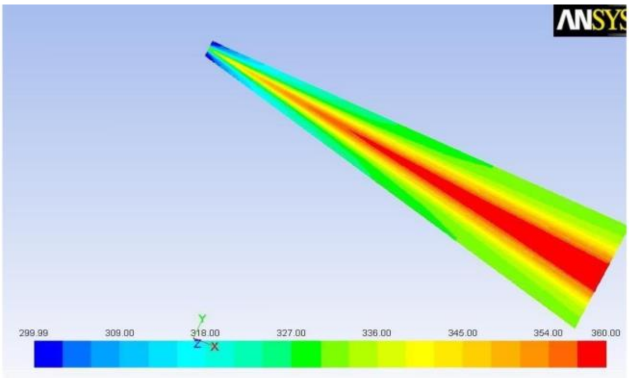

Figure 5.2 Result obtained

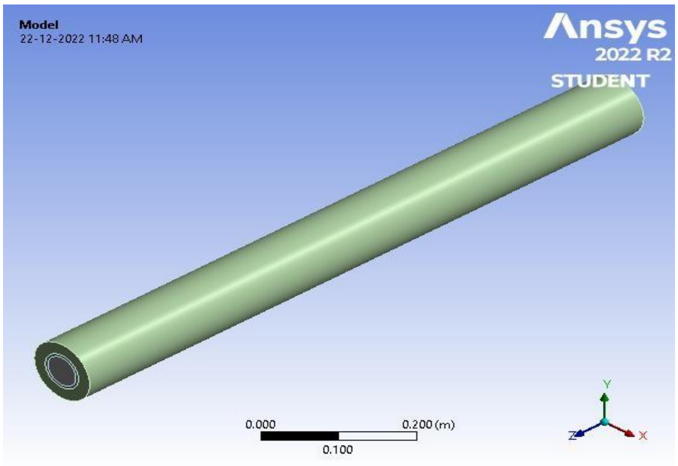

Figure 5.3 Heat exchanger design

6.RESULTS

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

7.CONCLUSION

A CFD analysis (ANSYS FLUENT 22.R2) was used for the numerical study of heat transfer characteristics of a double pipe heat exchanger for parallel flow and counter flow, and the results were then compared. The study showed that there is not much difference in the heat transfer within the error limits performances of the parallel-flow configuration and the counter-flow configuration. The simulation was carried out for water to water heat transfer characteristics and for same length and same diameter of tube input temperature for cold inlet 300k, for hot inlet 360k. We analysis that in counter flow heat exchanger there is high temperature difference in output streams (hot outlet, cold outlet).

When the fluid is passed through the double pipe heat exchanger in counter flow process the heat transfer is more compared parallel flow.

The temperature, pressure, velocity magnitude analysis for parallel flow and counter flow process done by the counters and compared the temperature, pressure, velocity magnitudes in parallel flow heat exchanger and counter flow heat exchanger.

ACKNOWLEDGEMENT

We are delighted to present the initial project report on ‘Disc Brake Caliper Hub’. Our heartfelt thanks go to our internal guide Mrs K. Deepika, for her unwavering assistance and guidance throughout the project. We are truly appreciative of their valuable suggestions, which proved to be extremely beneficial. We would also like to express our gratitude to the HOD, Dr. B. Vijaya Kumar, Guru Nanak Institute of Technology, Hyderabad, India, for their invaluable support and recommendations, which have been instrumental in the success of this project.

REFERENCES

[1] Concentric tube heat exchanger: operating principle with parallel flow. Art.

[2] White, Frank. Viscous Fluid Flow. 3rd ed. New York: The McGraw-Hill Companies, Inc., 2006.

[3] Conley, Nancy, Adeniyi Lawal, and Arun B. Mujumdar. ―An Assessment of the Accuracy of Numerical Solutions to the Gratz Problem.‖ Int. Comm. Heat Mass Transfer. Vol.12.

[4] Kays, William, Michael Crawford, and Bernhard Weigand. Convective Heat and Mass Transfer. 4th ed. New York: The McGraw-Hill Companies, Inc.,2005.

[5] Sellars J., M. Tribe’s, and J. Klein. ―Heat Transfer to Laminar Flow in a Round Tube or Flat Conduit—The Graetz Problem Extended.‖ The American Society of Mechanical Engineers. Paper No. 55-SA-66 AD-A280 848. New York. 1955.

[6] Subramanian, Shankar R. ―The Graetz Problem.‖ Web. 12 Apr. 2012. S

[7] K.S.Hwang,.K.Jang, S.U.S.Chio, Flow and convective heat transfer characteristics of water-based Al2O3 nanofluids in fully developed laminar flow regime, International Journal of Heat and Mass Transfer, 52 (2009) 193-199.

[8] V. Bianco, F. Chiacchio, O. Manca, S. Nardoni, Numerical investigation of nanofluids forced convection in circular tubes, Applied Thermal Engineering, 29 (2009) 3632 – 3642.

[9] M. Akbari, N. Galanos, A. Behzadmehr, Comparative analysis of singleand two-phase models for CFD studies of nanofluid heat transfer, International Journal of Thermal Sciences, 50 (2011) 1343 – 1354.

[10] Valko, Peter P. ―Solution of the Graetz-Brinkman Problem with the Laplace Transform Galerkin Method.‖ International Journal of Heat and Mass Transfer 48. 2005.

[11] Blackwell, B.F. ―Numerical Results for the Solution of the Graetz Problem for a Bingham Plastic in Laminar Tube Flow with Constant Wall Temperature.‖ Sandia Report. Aug. 1984. Encyclopædia Britannica Online. Web. 12 Apr. 2012.

[12] White, F.M., Fluid Mechanics, 3rd edition. Mc- Graw Hill, 1994- 2792

[13] Numerical study of heat transfer in a finned double pipe heat exchanger, Shiva Kumar, Vasudev Karanth, Krishna Murthy Dept of Mechanical Engg, MIT, Manipal, India (October 13 2014)