e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:05/Issue:06/June-2023 Impact Factor- 7.868 www.irjmets.com

CFD ANALYSIS OF DOUBLE PIPE HEAT EXCHANGER

Soham S. Kumbhar*[1], Prof. U. N. Patil*[2], Yash P. Patil*[3], Abhishek A. Shelake*[4], Kishor T. Dhere*[5], Piyush N. Shinde*[6]

*[1,3,4,5,6]Student, Department of Mechanical Engineering, D. Y. Patil College of Engineering & Technology, Kasaba Bawada, Kolhapur, Maharashtra, India

*[2]Assistant Professor, Department of Mechanical Engineering, D. Y. Patil College of Engineering & Technology, Kasaba Bawada, Kolhapur, Maharashtra, India

DOI : https://www.doi.org/10.56726/IRJMETS41354

ABSTRACT

While designing double pipe heat exchanger we face variety of problems because of the lack of experimental data available regarding the behavior of fluid flow in double pipe & also in case of heat transfer data, which is not in the case in shell & tube 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 covered the CFD analysis of double pipe parallel flow type heat exchanger. The materials for the study were decided, fluid taken was water and materials for the pipes were taken as Copper for inner pipe and Galvanized Iron [G. I.] for their better conducting properties. The CFD analysis was performed on ANSYS STUDNT 2022 R2 software using FLUID FLOW (Fluent).

Keywords: Analysis, double pipe, parallel flow, copper, galvanized iron, ANSYS STUDENT R2, CFD Analysis).

I. INTRODUCTION

Heat Exchange between flowing fluids is one of the most important physical process and a variety of heat exchangers are used in different types of installation, as in process industries, power plants, food processing, refrigeration, etc. 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 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. Advantages of double pipe heat exchanger include ease of cleaning and maintenance and usage under severe fouling conditions. Fluids at high pressure can also be used. Limitations include difficulty in the cleaning of tubes due to fouling and a shell and tube heat exchanger of the same design is a better way for heat transfer.

II. CFD DYNAMICS AND FLUENT

Computational fluid dynamics essentially means computational transport phenomenon which involves fluid dynamics, heat and mass transfer or any phenomenon involving the transport phenomenon. Computational fluid is applied in many fields of science like aerospace, biomedical, and chemical industry, automobile, electronics, marine engineering, material processing, turbo machines, etc. Fluent is a CFD code used for flow modelling applications. Fluent can analyze the given domain with boundary conditions and solve the governing equations for the flow given the different parameters. 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.

III. GEOMETRY OF THE HEAT EXCHANGER

Table 1. Geometric parameters

SN.	PARAMETER	DESCRIPTION
1	Inner dia. of inner tube	14.00 mm
2	Outer dia. of inner tube	16.00 mm
3	Inner dia. of outer tube	28.50 mm
4	Outer dia. of outer tube	32.50 mm
5	Length of both tubes	1000.00 mm

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:05/Issue:06/June-2023 Impact Factor- 7.868 www.irjmets.com

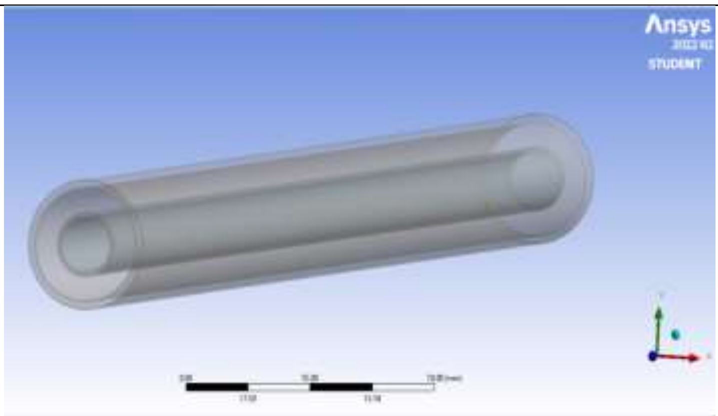


Figure 1: Geometry of double pipe heat exchanger

Named Selections - The different surfaces of the solid are named as per required inlets and outlets for inner and outer fluids.

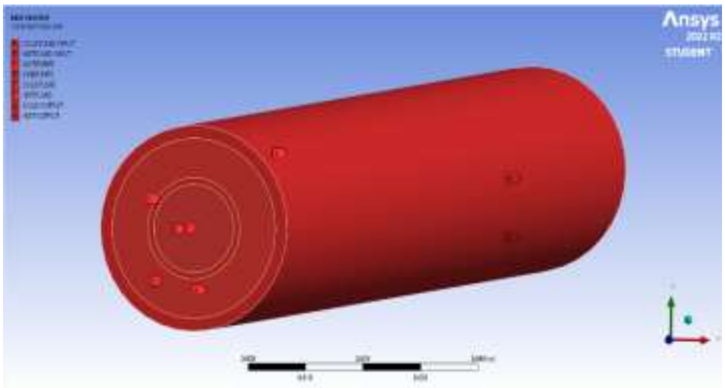


Figure 2: Named selections for the geometry

IV. MESH

Table 2. Mesh report

Domain	Nodes	Elements
Cold fluid	34104	24300
Hot fluid	31262	27540
Inner pipe	17864	9720
Outer pipe	22736	11745
Total	105966	73305

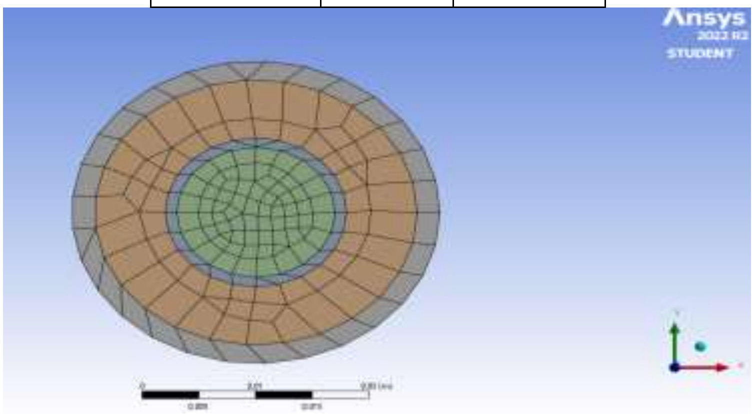


Figure 3: Meshing obtained

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:05/Issue:06/June-2023 Impact Factor- 7.868 www.irjmets.com

V. SETUP AND SOLUTION

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 steady state.

Models - Energy is set to ON positions. Viscous model is selected as "k-Ɛ model" and it is kept as realizable.

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

Cell zone conditions - Inner and outer fluid assigned as water and inner and outer solid pipe assigned as material copper and galvanized iron respectively.

Wall treatment – near wall treatment is set as enhanced wall treatment

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.

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 accurate result. For this reason residuals are given as per the table 4 that follows.

Table 3. Boundary conditions for parallel flow

Sr. No.	Boundary Conditions type	Mass flow rate (kg/sec)	Temperature (kelvin)
Hot inlet	Mass flow inlet	0.03	316.00K
Hot outlet	Pressure outlet	-	-
Cold inlet	Mass flow inlet	0.0231	301.00K
Cold outlet	Pressure outlet	-	-

Table 4. Measure of convergence

Variable	Residual
X – Velocity	10^{-6}
Y – Velocity	10^{-6}
Z - Velocity	10^{-6}
Continuity	10^{-6}
Turbulent Kinetic Energy	10^{-6}
Energy	10^{-6}

Run Calculation - The number of iteration is set to 250 and the solution is calculated.

VI. RESULTS AND DISCUSSION

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

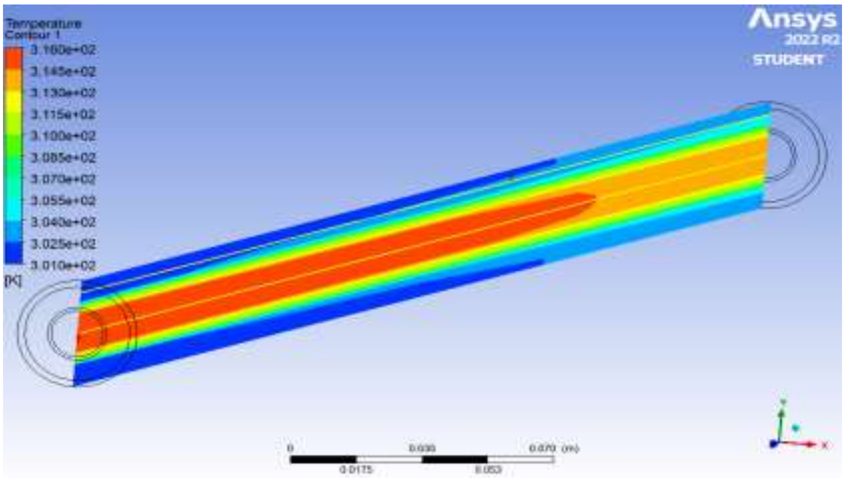
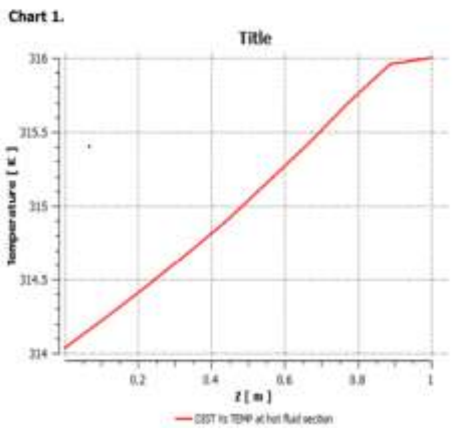


Figure 4: Contours of static temperature obtained in parallel flow

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:05/Issue:06/June-2023 Impact Factor- 7.868 www.irjmets.com

Charts – Distance Vs Temperature charts were generated.



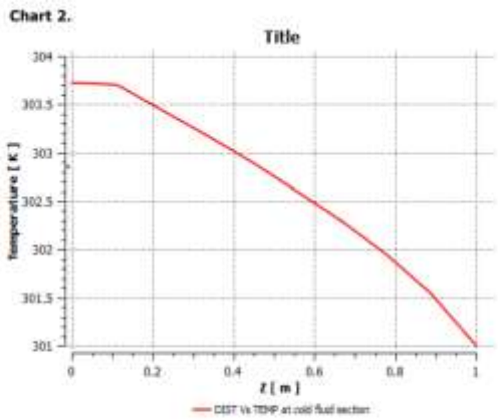


Figure 5: For hot fluid **Figure 6:** For cold fluid

VII. REPORT

Table 5: Temperature at inlet and outlet

Average Facet Values	In kelvin
Cold input	301.00
Cold output	304.39
Hot input	316.00
Hot output	313.83

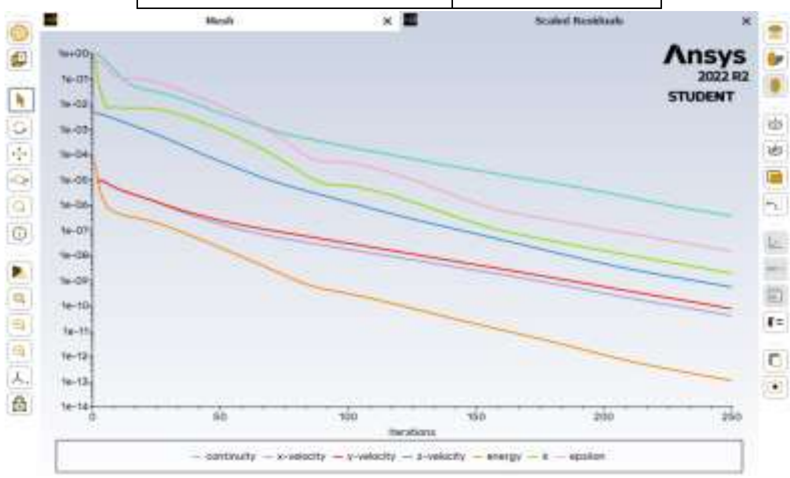


Figure 7: Variation of velocity, temperature, energy etc.

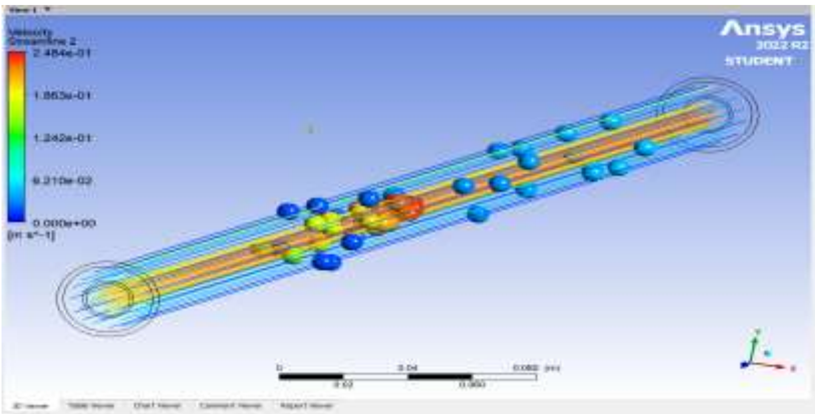


Figure 8: Cold and hot fluid streamlines

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
(Peer-Reviewed, Open Access, Fully Refereed International Journal)
Volume:05/Issue:06/June-2023 Impact Factor- 7.868 www.irjmets.com

VIII. CONCLUSION

A CFD package ANSYS FLUID FLOW (Fluent) was used for the numerical study of heat transfer characteristics of a double pipe heat exchanger for parallel flow, and the results were determined. The study showed the heat transfer is within the error limits performance of parallel-flow configuration. The simulation was carried 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 301K, and for hot inlet 316K. The temperatures obtained at outlet are 304.39K as cold water output and 313.83K as hot water output.

For the given design and length of heat exchanger heat transfer enhancement in a double pipe heat exchanger is possible achieved by several methods. Active methods involve some external input for the enhancement of the heat transfer like induced vibrations, injections and the passive method without the simulation by external power such as surface coating, surface roughness.

IX. REFERENCE

[1] Shreyas Kotian, Nachiket Methekar, “Heat Transfer and Fluid Flow in Double Pipe Heat Exchanger” Asian Review of Mechanical Engineering ISSN: 2249-6289 Vol.9 No.2, 2020, www.trp.org.in

[2] Dhrubajyoti Bhattacherjee, “CFD Analysis of Double Pipe Counter Flow Heat Exchanger” International Journal of Engineering Research & Technology (IJERT) (Oct, 2020) ISSN: 2278-0181, http://www.ijert.org/

[3] https://www.youtube.com/playlist?list=PLQMtm0_chcLx4NoPbUFkqLHoOomntN0Gr

[4] https://www.ansys.com/en-in/academic/students/ansys-student

[5] https://www.ansys.com/en-in/products/fluids/ansys-fluent

[6] ANSYS FLUENT manual