International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 09 Issue: 04 | April - 2025 SJIF Rating: 8.586 ISSN: 2582-3930

Design And Fabrication of a Helical Coil Heat Exchanger with Result Validation with CFD

[1]Shubham Pal, [2]Sneha Andade, [3]Chirag Patil, [4]Vaibhav Patil

[1]*Department of Mechanical Engineering, Alamuri Ratnamala Institute of Engineering and Technology, Shahapur*

---***---

Abstract-. Heat exchangers has been widely used in power plants and industries for heating or cooling applications. Heat is transferred from one medium to another medium by using heat exchangers. Helical coil heat exchangers with shell configurations are designed. The configurations has a copper helical coil. Designing of heat exchangers is done with counter flow And Parallel flow arrangement by using CATIA V5 R2015. The performance of both the configurations are analysed and compared by using Fluid flow (Fluent) in ANSYS WORKBENCH 18.1for CFD simulations. Further the helical tube materials Copper which are widely used in heat exchangers are also analysed in shell with core configuration and heat transfer rates are also compared using CFD (Fluent) analysis

Keywords: CFD Analysis, Corrugated Tube Heat Exchanger, Helical Coil

1.INTRODUCTION

The main objective of heat exchangers is to enhance the heat transfer between two fluids. The helical coil heat exchanger is designed with a helix angle of 13° which makes it compact and the heat transfer is greatly improved when compared to straight tubes. Due to the curved geometry of helical coil a centrifugal force is induced on the moving fluid. This results in development of a secondary flow. This secondary flow is the reason for increased heat transfer rates in helical coils.

The shell configurations are designed one core these configurations has a common material copper as inner tube material. The shell and tube fluids is water flowing in a counter flow arrangement. Performance of both the configurations are analysed.

Heat exchangers are essential devices used in various industries to transfer thermal energy between two fluids without mixing them. One efficient and compact type is the helical coil heat exchanger, which consists of a coiled tube typically submerged in a fluid-filled tank.

Amitkumar S Putterwar, A.M.Andhare has designed and thermally evaluated helical coil heat exchanger experimentally in a counter flow arrangement and how the heat transfer is improved due to secondary flow development [1]. Mr. M.D.Rajkamal et al has studied the use of POCO HTC graphite, ASTM SA 179 carbon steel and copper as inner tube materials in a shell with no core configuration using Ansys CFX 15.0 software and the results were compared [2]. Devendra Borse, Jayesh V. Bute has studied the helical coil heat exchanger on the basis of Dean Number, heat transfer coefficient and

Reynolds number and it infers that helical coiled tube heat exchanger is a promising modification over conventional type of heat exchanger [3]. Sreejith K, Jaivin A Varghese et al experimentally found that the effectiveness of helical coil heat exchanger is found to be higher than that of the straight tube heat exchanger for all the inlet temperatures of water. They have studied the variation of effectiveness with inlet temperature of hot water for both helical coil and straight tube heat exchangers. From the results obtained they have concluded that helical coil heat exchanger is having better effectiveness than straight tube heat exchanger [4]. Subin Michael has studied the changes in temperature profiles for different tube materials and the effectiveness of heat exchangers are calculated. CFD simulations are done for different industrial materials [5]. Shiva Kumara and K.Vasudev Karanth has studied the performance helical coiled tubular heat exchanger used for cooling water under constant wall temperature conditions by CFD simulations. CFD results of helical coiled tubular heat exchanger are compared with the results of straight tubular heat exchanger of the same length under identical operating conditions. Results showed increase in heat transfer and increase in nusselt number when compared to straight tubular heat exchanger [6].

2. LITERATURE REVIEW

In recent years, extensive research has proven that nanofluids are superior as a heat transfer agent over conventional fluid. Numerous aspects are still ongoing and need to be addressed in this field. In this section, a literature review is presented to facilitate the discussion on the experimental work the conducted in this study.

[1] Masuda et al. (1993) presented the structure of liquid suspended of nano-sized particles it was Choi (1995) who proposed the name of nanofluid for the first time. Ever since Choi published the first findings in NFs studies, there have been several other works has been done to the improvement of HT up to 20% by using densely distribution of nanoparticles in NFs. Efforts were carried out for better comprehension of changes in heat transfer coefficient in heat exchangers. Heat transfer coefficient of NFs with very low particle volume% is much higher as referred to the base fluid. On the other hand, low changes in friction coefficient and fluid viscosity in NFs have been reported.

[2] Xuan and Roetzel et al. has noticed an increase in energy transfer rate in their investigation on random motion of nanoparticles in NF. An experimental study on the convectional and flow characteristics of water-Cu NF through a straight pipe with constant thermal flow under laminar and turbulent regimes has been reported. Nanoparticles of Cu with less than 100 nm diameter were employed. The results show that Nano-suspended particles substantially improved the performance of conventional base fluid HT. The volume fraction of base fluid in NF fits well with that of water. Pak and Cho (1998) found in their experiment the turbulent forced convection heat transfer of Al_2O_3 water is higher than TiO_2 /water nanofluids inside a circular tube. Li and Xuan (2002) concluded that in laminar and turbulent flow regime in forced convection, the heat transfer coefficient of Cu/water nanofluids flowing inside a uniformly heated tube remarkably increased compared to that of pure water.

[3] Lotfi et al. Have compared the single-phase with the Mixture and Eulerian two-phase models for the forced convection flow of Al_2O_3/Water nanofluid with temperature independent properties. Also, they have compared the Nusselt number predictions for a 1% value concentration of nanoparticles with several correlations and one set of experimental values. They have also studied the effect of volume concentration on the wall temperature. Their results showed that the Mixture model is more precise than the other two models.

[4] Effect of Nanoparticles on Thermal Efficiency of Double Tube Heat Exchanger Reza Aghayari et al. did the experiments to find an Overall Heat Transfer Coefficient of Nano Fluids (OHTCNF)in heat exchangers and other relevant effective parameters. An improvement in Heat Transfer (HT)and OHTCNF containing Nano-aluminum oxide with ca. 20 nm particle size and particular volume fraction in the range of 0.001-0.002 was reported. The effects of temperature and concentration of nanoparticles on HT variation as well as Overall Heat Transfer Coefficient (OHTC) in a countercurrent double tube heat exchanger with turbulent flow have been studied. The experimental results fig:1 shows a remarkable 8%-10%riseinthe mean HT and the OHTC. In general, there are three mechanisms to improve heat transfer by introducing nanoparticles into the base fluid. Nano-particles benefit higher heat transfer rate; therefore, as nanoparticle concentration in the base fluid increases the heat transfer rate increases accordingly. The collisions occur between nanoparticles and the base fluid molecules on the one hand and the impacts of the particles to the heat exchanger wall on the other hand result in an energy increase. The friction between the wall and fluid increases if NFs are dealt with and, therefore, heat transfer improves.

[5] The effect of nanoparticles on the heat transfer properties of drilling fluids Hassani et al. has been found that the velocity and temperature have an important effect on the thermal property of mud. The thermal performance factor for all the cases is greater than base mud (5-22% for 0.01-2 wt.% nano-material) and convection results showed that the maximum thermal performance was found for the hybrid of CNT-silica nano-particle in higher Reynolds number. The heat transfer enhancement in 4200 Reynolds number, is 31%.

[6] CFD accurately predicted heat transfer and Nusselt number for a three-dimensional tube in tube heat exchanger. Similarly, CFD provided good

agreement with analytical and experimental results for a prototype heat exchanger. Where the experimental and CFD flows were similar, a good correlation for friction was found between the CFD and experimental results. It was concluded that CFD is valuable tool in heat exchanger design.

[7] The convective heat transfer and pressure drop characteristics

of Al_2O_3–water nanofluid flowing in helically coiled tube-in-tube heat exchangers were numerically investigated. The effects of nanoparticle volume concentrations and curvature ratio on the heat transfer and pressure drop characteristics were determined. Based onto the obtained results, the following conclusions can be stated:

(1) The 3D realizable k–e (RKE) model with enhanced wall treatment is robust and sufficient to simulate the turbulent flow and heat transfer of water and nanofluids in CTITHEs.

(2) The criteria for comparison of the heat transfer coefficient and pressure drop between base fluid and nanofluids should be carefully selected. Usually in the literature the comparisons are based on Re, this might be illusory as the higher heat transfer coefficient for the nanofluids is not because

of better nanofluids performance, but due to higher volume flow rate.

(3) When the comparison is at the same Re, the combined effects of the increased heat transfer capabilities of nanofluids and the secondary flow can be used as a compound passive approach to maximize the effectiveness of a heat exchanger and concurrently reduce the size of the heat exchanger.

(4) Also, when the comparison is at the same Re or Dn, the heat transfer coefficient increases by increasing the coil diameter and nanoparticles volume concentration. Moreover, the friction factor increases with the increase in the curvature ratio and almost there is no pressure drop penalty with increasing the nanoparticles volume concentration up to 2%.

[8] Heat transfer enhancement at least 1.4 times of water in straight tubes was observed in the heat transfer tests of SPE98330 (1500 ppm) solution in the fluted tube-in-tube heat exchanger with only modest pressure drop penalties These are very encouraging results. The super-ordered micelle structure of this solution may experience a breakdown by the shear stress induced by the spirally fluted tube. This was not found in the experiments for Ethoquad T13-50/NaSal solution in the same tube and indicates that the Ethoquad T13- 50/NaSal (5 mM/8.75 mM) solution has a stronger microstructure than the SPE98330 (1500 ppm) solution, which also degraded significantly with continuous circulation.

[9] A numerical study has been performed to investigate heat transfer and friction factor of fluid flow in the annulus region of a tube in tube helical heat exchanger for the laminar regime at different Dean Numbers. The behavior of the overall heat transfer coefficient under the influence of different cross-sections of the inner pipe of the heat exchanger revealed that effectiveness and overall heat transfer coefficient was strongly affected with Dean Number. The use of different geometry of the inner pipe of the tube in tube helical heat exchanger causes a higher friction factor at low Dean Number. The Nusselt Number for Geometry B to E is greater than that of a circular tube and was found to increase with Dean Number. The use of geometry E increases the Nusselt number and friction factor by 17.05% and 15% respectively at a Dean number of 400 as compared with a circular tube. Nusselt number of Geometry B increases by 13.73% as compared to Geometry A. It is observed that the increase in Nusselt number from Geometry B to C is 1.45% and geometry C to E is 3.24%.

[10] The hydrodynamics and heat transfer characteristics of compressed air with turbulent flow in the tube in tube helical heat exchanger were investigated for the first time at a pilot plant scale. The friction factor values for compressed air were compared with values reported in the literature for ambient conditions. In the literature, most of the data on heat transfer coefficients in coiled tubes are reported with wall boundary conditions of either constant heat flux or constant wall temperature. In the present study, physically realistic boundary conditions of fluid-to-fluid heat transfer were used. The increase of outer tube Nusselt number was found to be significant in tube in tube heat exchanger as compared to coiled tube and straight tube because of the presence of semicircular baffles. The experimental results from present work agreed with the CFD modelling results of Kumar et al.26 The results from the experiments have been summarized in form of new correlations for friction factor and Nusselt number for the inner and outer tubes of the tube in tube helical heat exchanger.

[11] Extended performance evaluation criteria equations have been used to assess the thermodynamic efficiency of some techniques to enhance heat trans fer in the annulus of tube-in-tube heat exchangers, such as: angled spiraling tape inserts, a round tube inside a twisted square tube and spiraled tube inside the annulus. The heat transfer enhancement in the shell can be supplemented by heat transfer augmentation in tubes using twisted tape inserts or micro-finned tubes. The effect of the thermal resistance of the con dens ing refrigerant could also be taken into consideration. The evaluation of the performance of each technique has been made based on the first and second law analyses, considering some design or operational constraints. The results show that in most of the cases considered, the angled spiraling tube insert technique is the most efficient.

3. MATERIALS AND METHODS

1. Materials

The materials selected for the fabrication and simulation of the helical coil heat exchanger are based on thermal conductivity, corrosion resistance, mechanical strength, and ease of manufacturing.

Coil Material Reason: High thermal conductivity (≈ 385 W/m·K for copper), excellent corrosion resistance, and good formability.

Tube Dimensions*: Outer Diameter = 10 mm, Wall Thickness = 3 mm, Number of Turns = 6

Working Fluids

- Hot Fluid: Water (60°C – 90°C)

- Cold Fluid: Water or Ethylene Glycol solution (ambient to 30°C)

- Reason: Readily available and safe for lab-scale testing.

2. Design Parameters

Coil Type: Helical coil with constant pitch

- Pitch: 7 mm

- Tube Inner Diameter: 8 mm

- Number of Turns: 6

- Flow Rate (Hot): 0.2–0.5 L/min

- Flow Rate (Cold): 0.2–0.5 L/min

-Orientation: Vertical helical coil inside cylindrical shell

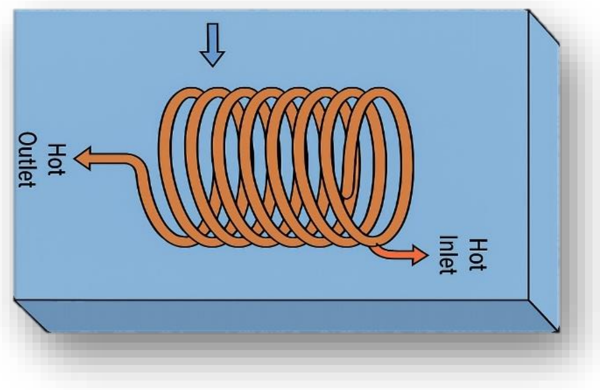

Fig 3.1 Helical Coil.

3. Fabrication Process

1. Coil Bending: A copper or stainless-steel tube is wound into a helical shape using a mandrel and coil-forming jig.

2. Shell Preparation: Mild steel pipe is cut and drilled for inlet/outlet ports.

3. Assembly: The coil is inserted into the shell, and the end connections are brazed or welded.

4. Sealing and Insulation: Leak-proof testing is conducted, and insulation is added to minimize heat loss.

4. ANSYS Simulation (CFD Analysis)

Geometry Creation

- Created using ANSYS Design Modeler.

- A 3D model of the shell and helical coil is constructed with accurate dimensions.

Meshing

- Tool Used: ANSYS Meshing

- Mesh Type: Tetrahedral for coil and shell

- Element Size: Optimized between 1 mm to 3 mm for accurate boundary layer capture

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 09 Issue: 04 | April - 2025 SJIF Rating: 8.586 ISSN: 2582-3930

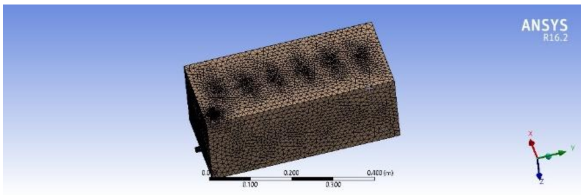

Fig.3.2 Tank with Meshing

-Inlets: Mass flow rate and temperature specified for both fluids.

- Outlets: Pressure outlet condition.

- Wall Conditions: No-slip condition applied to all walls.

4.4 Solver Setup

- Software: ANSYS Fluent

- Model: Energy equation activated; laminar or k-epsilon turbulence model selected based

Fig 3.2 Schematic of experimental set up

4. CFD SIMULATION

4.1 Introduction

Computational fluid dynamics (CFD) is a computer-based simulation method for analysing fluid flow, heat transfer, and related phenomena such as chemical reactions. This dissertation uses CFD for analysis of flow and heat transfer. It will be advantageous to use CFD over traditional experimental-based analyses, since experiments have a cost directly proportional to the number of configurations desired for testing, unlike with CFD, where large amounts of results can be produced at practically no added expense. In this way, parametric studies to optimize equipment are very inexpensive with CFD when compared to experiments.

4.2 CFD Analysis Process

To perform a CFD analysis, the analyst will state the problem and use scientific Knowledge to express it mathematically. Then the CFD software package will embody this knowledge and expresses the stated problem in scientific terms. Finally, the computer will perform the calculations dictated by CFD software and the analyst will inspect and interpret their results. In principle, three different major tasks should be done to perform a CFD simulation.

4.3 Problem Identification

The final goal is to develop CFD-methods for realistic prediction of the overall heat transfer coefficient in a passage containing in-line array of pin fins, to improve the heat transfer efficiency. Therefore, generic flow cases with three different fin geometries that have the typical flow-heat transfer characteristics are investigated. The focus of this work is on the investigation of the effects of fin morphology in prediction of the flow and heat transfer in a typical heat exchanger passage. Also, the study of the underlying physics of the flow-heat transfer processes in these cases is included.

4.4 Model Development in ANSYS

The ANSYS Design Modeler is a gateway to geometry coping with for an ANSYS analysis or we can import the model file of other software like CAD, SOLIDWORKS. The geometry consists the physics or physical structure. In this we have developed our model in SOLIDWORKS 2017 and then imported it to ANSYS software. the reason for selecting SOLIDWORKS is that we are familiar with it and secondly for ease in making small protrusions on the tube which is very tough to draw in ANSYS.

Create a Geometry - All engineering simulations start with geometry to represent the design, be it a solid component of a structural analysis or the air volume of a fluid or electromagnetic field. The engineer either has geometry that has been created in a SOLIDWORKS system or builds the geometry from scratch. The ANSYS Design Modeler is a gateway to geometry handling for an ANSYS analysis. Geometry created using ANSYS Design Modeler software which is specifically designed for the creation and preparation of geometry for simulation. In engineering simulations, the geometry includes details not needed for simulation. Only the physics involved is to be included, simulating such a fully detailed model.

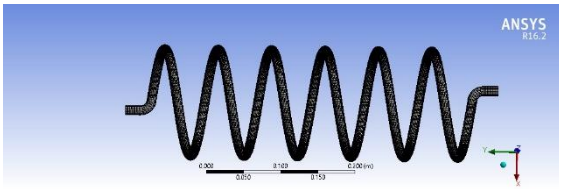

Fig. 4.1 Helical Coil

4.5 Selection of Mesh and Reason

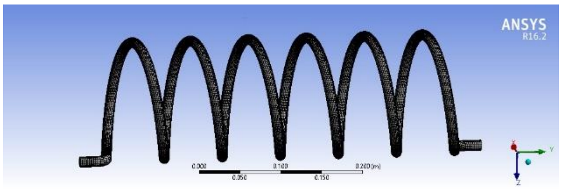

Fig. 4.2 Meshing with Coil

Tetrahedron- A tetrahedron has 4 vertices, 6 edges, and is bounded by 4 triangular faces. In most cases a tetrahedral volume mesh can be generated automatically. When geometries are complex or the range of length scales of the flow is large, a triangular/tetrahedral mesh can be created with far fewer cells than the equivalent mesh consisting of quadrilateral/hexahedral elements. This is because a triangular/tetrahedral mesh allows clustering of cells in selected regions of the flow domain. Structured quadrilateral/hexahedral meshes will generally force cells to be placed in regions where they are not needed. Unstructured quadrilateral/hexahedral meshes offer many of the advantages of triangular/tetrahedral meshes for moderately-complex geometries.

Choosing a Turbulence Model –

The eight RANS turbulence models differ in how they model the flow close to walls, the number of additional variables solved for, and what these variables represent. All of these models augment the Navier-Stokes equations with an additional turbulence eddy viscosity term, but they differ in how it is computed.

k-ε

The k-ε model solves for two variables: k, the turbulence kinetic energy; and ε (epsilon), the rate of dissipation of turbulence kinetic energy. Wall functions are used in this model, so the flow in the buffer region is not simulated. The k-ε model has historically been very popular for industrial applications due to its good convergence rate and relatively low memory requirements. It does not very accurately compute flow fields that exhibit adverse pressure gradients, strong curvature to the flow, or jet flow. It does perform well for external flow problems around complex geometries. For example, the k-ε model can be used to solve for the airflow around a bluff body. The turbulence models listed below are all more nonlinear than the k-ε model and they can often be difficult to converge unless a good initial guess is provided. The k-ε model can be used to provide a good initial guess. Just solve the model using the k-ε model and then use the new Generate New Turbulence Interface functionality, available in the CFD Module with COMSOL Multiphysics version 5.3.

k-ω

The k-ω model is like the k-ε model, but it solves for ω (omega) — the specific rate of dissipation of kinetic energy. It is a low Reynolds number model, but it can also be used in conjunction with wall functions. It is more nonlinear, and thereby more difficult to converge than the k-ε model, and it is quite sensitive to the initial guess of the solution. The k-ω model is useful in many cases where the k-ε model is not accurate, such as internal flows, flows that exhibit strong curvature, separated flows, and jets. A good example of internal flow is flow through a pipe bend.

Low Reynolds Number k-ε

The low Reynolds number k-ε model is like the k-ε model, but does not need wall functions: it can solve for the flow everywhere. It is a logical extension of the k-ε model and shares many of its advantages, but generally requires a denser mesh; not only at walls, but everywhere its low Reynolds number properties kick in and dampen the turbulence. It can sometimes be useful to use the k-ε model to first compute a good initial condition for solving the low Reynolds number k-ε model. An alternative way is to use the automatic wall treatment and start with a coarse boundary layer mesh to get wall functions and then refine the boundary layer at the interesting walls to get the low Reynolds number models. The low Reynolds number k-ε model can compute lift and drag forces and heat fluxes can be modelled with higher accuracy compared to the k-ε model. It has **SST**

The SST model is a combination of the k-ε model in the free stream and the k-ω model near the walls. It is a low Reynolds number of model and kind of the "go to" model for industrial applications. It has similar resolution requirements to the k-ω model and the low Reynolds number k-ε model, but its formulation eliminates some weaknesses displayed by pure k-ω and k-ε models. In a tutorial model example, the SST model solves for flow over a NACA 0012 Air foil. The results are shown to compare well with experimental data.

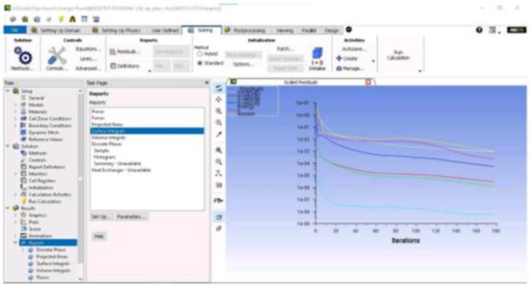

Fig.4.3 Iterations

5. RESULTS AND DISCUSSION

CFD Results

The Scale Residual is shown in fig. 6.5. Different colour lines represent the different equations and parameters and on X axis it shows number of iterations to perform the calculations. White line denotes the continuity equation, purple line denotes the X velocity. Dark blue line represents the Y velocity, parrot green colour line denotes energy and yellow line represents the position.

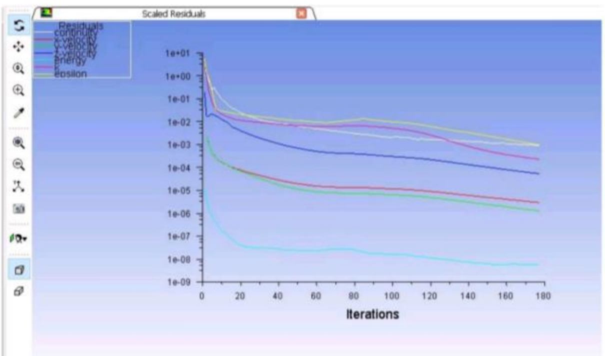

Fig. 5.1 Scale Residual

Velocity of hot and cold fluid In this simulation we have done analysis on tube in tube heat exchanger. Hot fluid is passing through the inner tube and Cold fluid is passing through the outer tube. The heat is transferred from hot fluid to cold fluid and hot fluid get cool down from 82 °C to 79°C and Cold fluid get heated from 36°C to 39°C. It means we will get temperature difference of 3°C for counter flow. For parallel flow this difference become less i.e. 2°C. By changing the dimension and materials we can get different values of the temperature difference. The velocity of both hot and cold fluid is shown in below fig. 5.2 and 5.3.

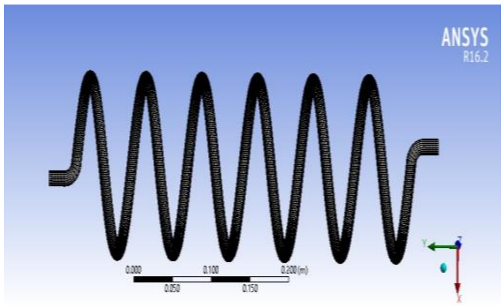

Fig. 5.2 Velocity of Hot and Cold Fluid

TEMPERATURE OF HOT AND COLD FLUID ;In this simulation we have done analysis on tube in tube heat exchanger. Hot fluid is passing through the inner tube and Cold fluid is passing through the outer tube. The temperature Contour is shown in fig. below by CFD analysis it shows that hot fluid temperature by red colour and cold fluid temperature by blue colour.

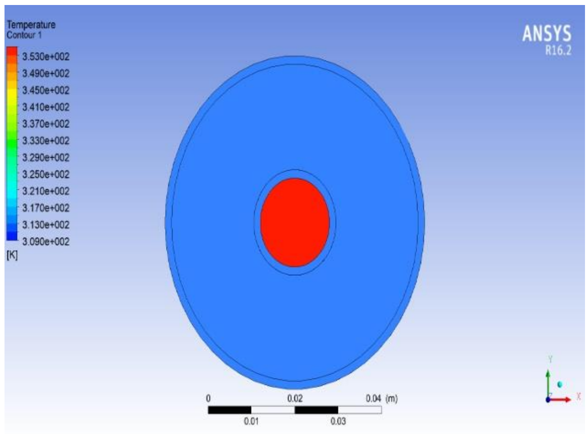

Fig. 5.3 Temperature Contour for Hot Fluid and Cold Fluid

6. CONCLUSION

The design and fabrication of a helical coil heat exchanger were successfully carried out, followed by detailed thermal and structural analysis using ANSYS. The helical coil configuration demonstrated superior heat transfer capabilities due to the enhanced turbulence and increased surface area offered by the coil geometry. The simulation results validated the design parameters, showing effective temperature gradients and stable thermal performance under the given boundary condition. The structural analysis in ANSYS further confirmed that the exchanger could withstand the expected pressure and thermal loads without significant deformation or failure, ensuring operational reliability. The fabricated model matched the design specifications, and initial performance tests indicated that the exchanger performs as predicted in the simulations Overall, the integration of simulation tools like ANSYS in the design process greatly improved the efficiency and accuracy of the development cycle, reducing the need for multiple physical prototypes. This project demonstrates the effectiveness of helical coil designs for compact and efficient heat transfer applications in various industries.

7. REFERENCES

1.Kakac,S., & Liu, H. (2002). Heat Exchangers: Selection, Rating, and Thermal Design (2nd ed.). CRC Press.

2. Holman, J. P. (2010). Heat Transfer (10th ed.). McGraw-Hill Education.

3. Naphon, P., & Wongwises, S. (2006). A review of flow and heat transfer characteristics in curved tubes. Renewable and Sustainable Energy Reviews, 10(5), 463–490.

4. Prabhanjan, D. G., Raghavan, G. S. V., & Rennie, T. J. (2002). Comparison of heat transfer rates between a straight tube heat exchanger and a helical coil heat exchanger. International Communications in Heat and Mass Transfer, 29(2), 185–191.

5. ANSYS, Inc. (2023). ANSYS Fluent Theory Guide. Available at: https://www.ansys.com

6. Ghorbani, N., Taherian, H., Gorji, M., & Mirgolbabaei, H. (2010). Experimental study of thermal performance in a helical coiled heat exchanger made from corrugated tube. Energy Conversion and Management, 51(3), 546–552.

7. Kumar, V. (2006). Design of compact heat exchangers for laminar flow. AIChE Journal, 52(5), 1733–1741.

8. Mahajan, A., & Kaushik, S. C. (2014). Performance enhancement of helical coil heat exchangers with non-circular tubes. Applied Thermal Engineering, 71(1), 456–464.

9. Ghoshdastidar P. S. (2011). Hydrodynamics and Heat Transfer in Fluid Flow Systems. CRC Press.

10. Shah, R. K., & London, A. L. (1978). Advanced Heat Transfer. Prentice Hall.

11. Lee, S. H., & You, S. S. (2006). The effect of coil diameter and length on the thermal-hydraulic performance of helical coil heat exchangers. Heat Transfer Engineering, 27(6), 38–48.

12. Kandpal, T. C., & Kumar, M. (2008). Energy Efficient Buildings: Design and Case Studies. Springer Science & Business Media.

13. D'Agaro, R., & Tesser, G. (2017). CFD study of a helical coil heat exchanger under turbulent flow conditions. Computers & Fluids, 145, 163–175.

14. Ranjan, R., & Chhabra, R. P. (2017). Design and analysis of a helical heat exchanger for enhanced heat transfer performance. International Journal of Heat and Mass Transfer, 104, 136–149.

15. Orf, L. H., & DeLiso, A. R. (2014). Experimental and computational analysis of heat transfer in helical coil heat exchangers. International Journal of Thermal Sciences, 85, 63–72.

16. Wawrzyniak, R., & Zdyb, J. (2015). Heat transfer analysis in a helical coil heat exchanger using CFD. Applied Thermal Engineering, 86, 196–204.

17. Zhang, L., Li, Z., & Yang, Y. (2016). Numerical study on the thermal performance of helical coil heat exchangers. Energy, 106, 1139–1149.

18. Lee, S. H., & You, S. S. (2015). Heat transfer and pressure drop characteristics of helical coil heat exchangers. International Journal of Heat and Mass Transfer, 82, 135–145.

19. Wang, L., & Xu, W. (2012). Investigation of heat transfer in helical coiled heat exchangers: a review. Renewable and Sustainable Energy Reviews, 16(5), 3645–3655.

20. Jindal, S., & Mathur, A. (2016). Performance analysis of helical coil heat exchangers under turbulent flow conditions. International Journal of Heat and Mass Transfer, 98, 105–113.