IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 11, April 2025
Impact Factor: 7.67

# Numerical Simulation of Flow and Thermal Performance of a Spiral Coil Heat Exchanger using ANSYS Fluent

**Sumit Ganguly, Kriss Kumar Pandit, Sumit Kumar Sharma, Deepak Kumar Rawani, Vikash Kumar Thakur, Chhotan Rajwar & Deepak Bauri**
Department of Mechanical Engineering
K. K. Polytechnic, Dhanbad

**Abstract:** *The objective of this project is to numerically model the flow and thermal performance of a spiral coil heat exchanger using ANSYS Fluent. Efficient, compact, and having the ability to process high-pressure fluids, spiral coil heat exchangers are popular. Computational Fluid Dynamics (CFD) is a preferred method of performance analysis since conventional analytical methods are not sufficient owing to their complex geometry. This paper employed velocity and pressure boundary conditions in the simulation and modeling of a spiral coil heat exchanger with inner and external fluid flows. The two fluids are separated by a copper pipe, namely hot water inside and cold water outside. With ordinary turbulence models, the simulation investigates pressure profiles, velocity fields, as well as temperature distribution in turbulent settings. The results provide information on the exchanger's thermal and flow behaviour by displaying precise contours of dynamic pressure, static temperature, and effective thermal conductivity.*

**Keywords:** ANSYS Fluent

## I. INTRODUCTION

Heat exchangers are integral components of thermal systems that are extensively used in industries such as HVAC, chemical processing, power generation, and refrigeration to effectively transfer thermal energy between two or more fluids. Due to its smaller size, enhanced heat transfer performance, and ability to process high-pressure fluids, spiral coil heat exchangers are more commonly used compared to other variations. In comparison with straight tubes, its geometry generates centrifugal forces that enhance the rates of convective heat transfer by promoting secondary flows, known as Dean vortices [1]. Spiral coil flow and temperature profiles are significantly influenced by the curvature of the tube. Because of its geometrical nature, spiral coils can be applied to applications where weight and area are limited but efficient heat transfer is required.It is difficult to analyze spiral coil heat exchangers using traditional analytical methods because they have a complex design. Consequently, their performance is increasingly simulated and optimized with Computational Fluid Dynamics (CFD) technologies like ANSYS Fluent [2].Apart from offering extensive analysis of pressure drops, thermal gradients, and local heat transfer coefficients, CFD simulations contribute to the visualization of temperature and velocity distributions. Curved geometries' flow behavior is well-represented by numerous turbulence models such as k-ε and k-ω SST [3]. These models are particularly useful in examining the impact of design parameters like fluid properties, coil pitch, tube size, and flow rate on performance.Earlier work has established that in laminar and turbulent regimes, spiral coils offer significant advantages over straight tubes. Curvature-promoted flow enhances heat transfer characteristics, as has been demonstrated experimentally and computationally by Naphon [1].Similarly, Patil and Vedula [4] have studied the thermal and hydraulic performance of spiral coils through CFD, pointing out the ways in which exchanger efficiency can be enhanced by modification to design.In this project, the heat transfer and flow characteristics of a spiral coil heat exchanger are numerically evaluated with the help of ANSYS Fluent. Some specific objectives include:
simulating the internal temperature and velocity fields of the coil under different operating conditions.

IJARSCT

International Journal of Advanced Research in Science, Communication and Technology

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

ISSN: 2581-9429

Volume 5, Issue 11, April 2025

Impact Factor: 7.67

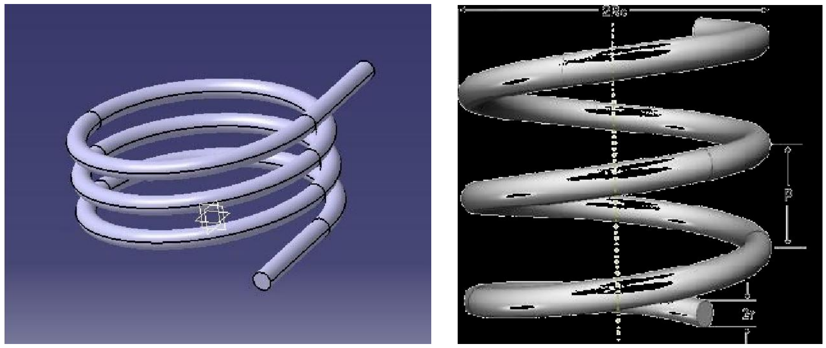

Fig:1:- Schematic diagram of helical coil heat exchanger

**Analysis**

We are taking velocity as the input and pressure as the output parameters. There will be two inlets and an outlet, respectively, since this is the reverse of the inner hot fluid flow and the outer cold fluid flow. The two flows are divided by a copper-made pipe. These are the details of each boundary condition. Hot water is employed for the inner fluid and cold water for the outside fluid.

| Location | Boundary condition type | Velocity magnitude | Turbulent kinetic Energy | Turbulent dissipation rate | Temperature |
|---|---|---|---|---|---|
| Inner inlet | Velocity Inlet | 1.6 m/s | 0.01 | 0.1 | 333 K |
| Inner inlet | Pressure Outlet | - | - | - | - |
| Outer inlet | Velocity Inlet | 1.5m/s | 0.01 | 0.1 | 283 K |
| Outer inlet | Pressure Outlet | - | - | - | - |

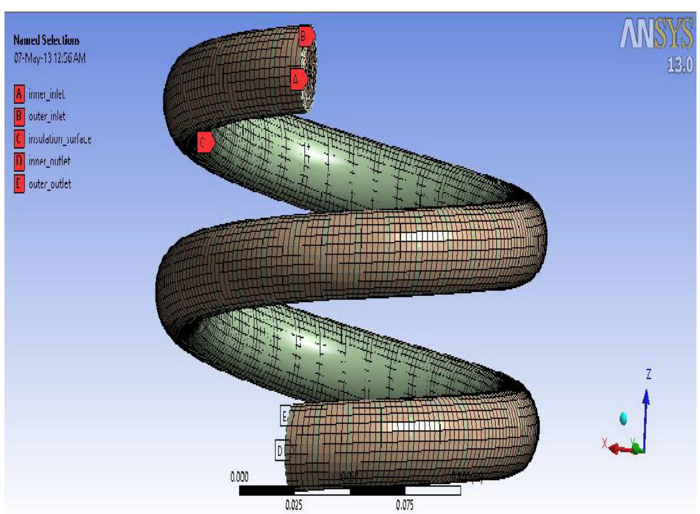


Fig 2:- Ansys diagram of helical coil heat exchanger

Diameter of inner inlet= 0.58 inch

Diameter of inner pipe= 0.67 inch
Diameter of outer inlet= 0.82 inch
Diameter of outer pipe= 0.91 inch
Diameter of coil= 5.59 inch
Density is taken in y direction = 9.80 m/s2

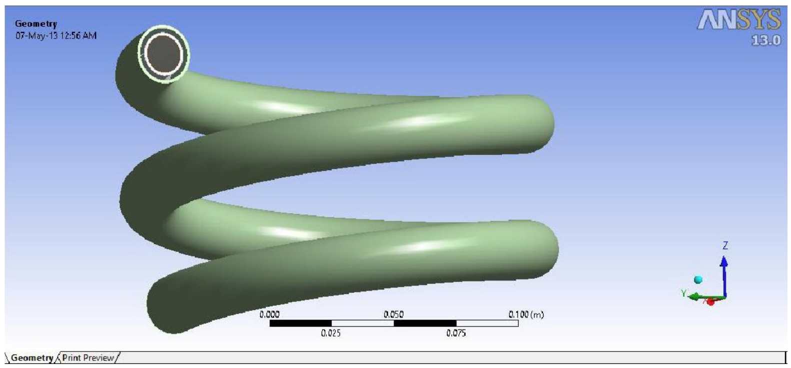


Fig 3- Ansys diagram (Solid) of helical coil heat exchanger

## II. RESULT & DISCUSSION

The corresponding contours show the distribution of temperature, pressure, and velocity along the heat exchanger.

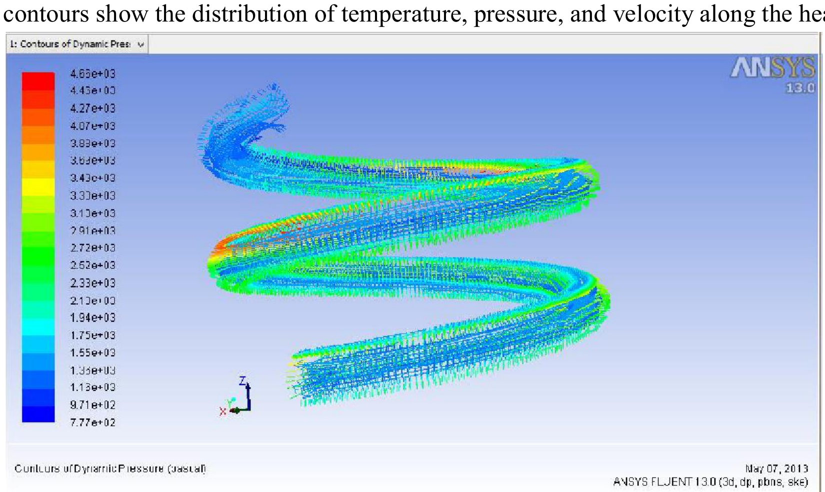


Fig 4- Contours of Dynamic Pressure in Pascal

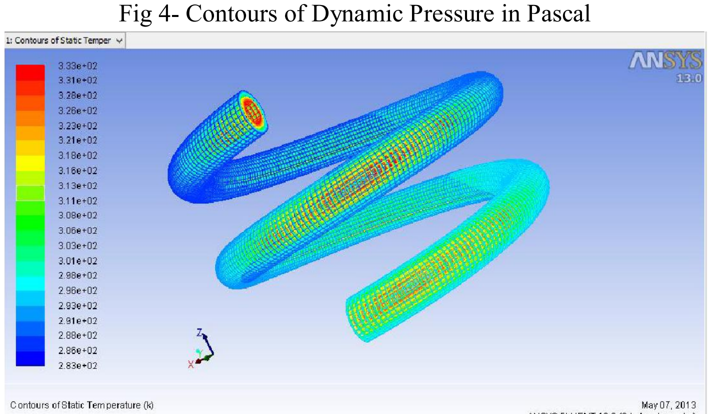


Fig 5- Contours of static temperature

IJARSCT
International Journal of Advanced Research in Science, Communication and Technology
International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal
ISSN: 2581-9429
Volume 5, Issue 11, April 2025
Impact Factor: 7.67

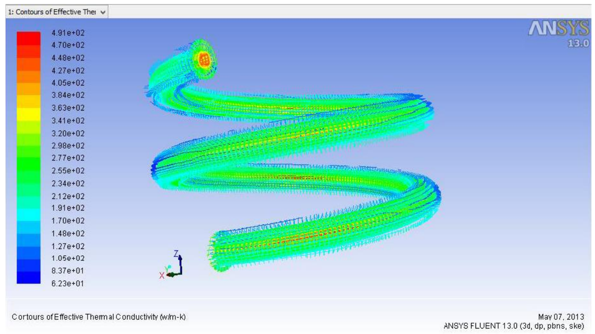


Fig 6- Contours of effective thermal conductivity

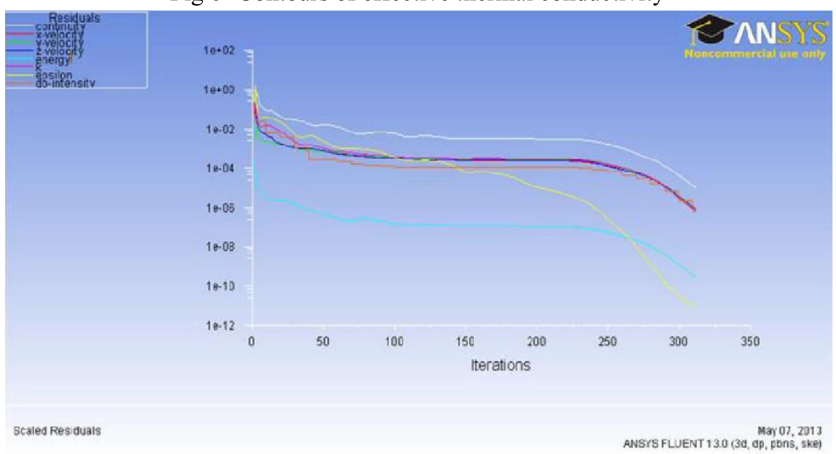


Fig 7- Scaled residuals

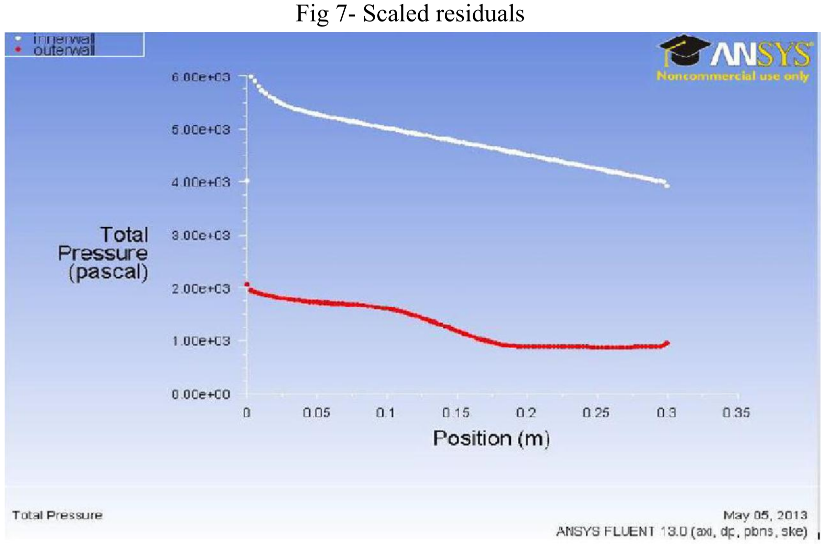


Fig 8- Total Pressure Plot for Innerwall and Outerwall

## III. CONCLUSION

The thermal and flow characteristics of a spiral coil heat exchanger were successfully evaluated by the CFD simulation conducted using ANSYS Fluent. The simulation results confirmed the enhanced heat transfer performance attributed to

IJARSCT

International Journal of Advanced Research in Science, Communication and Technology

International Open-Access, Double-Blind, Peer-Reviewed, Refereed, Multidisciplinary Online Journal

ISSN: 2581-9429

Volume 5, Issue 11, April 2025

Impact Factor: 7.67

the spiral configuration by demonstrating the effective distribution of pressure and temperature across the exchanger. The complex flow behaviour, such as the creation of secondary flows that enhance convective heat transfer, was accurately simulated by the employment of turbulent models. In general, the research illustrated how important computer modelling is to enhancing the operation and designs of heat exchangers. To achieve greater efficiency and applicability to a variety of industrial systems, future research can test various coil sizes, flow rates, and material properties.

## REFERENCES

[1] P. Naphon, "Thermal performance and pressure drop of the helical-coil heat exchangers with and without helically crimped fins," *Int. Commun. Heat Mass Transfer*, vol. 34, no. 3, pp. 321–330, Mar. 2007, doi: 10.1016/j.icheatmasstransfer.2006.11.004.

[2] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, 2nd ed. Harlow, UK: Pearson Education, 2007.

[3] T. L. Bergman, A. S. Lavine, F. P. Incropera, and D. P. Dewitt, *Fundamentals of Heat and Mass Transfer*, 7th ed. Hoboken, NJ, USA: Wiley, 2011.

[4] A. V. Patil and R. P. Vedula, "CFD analysis of spiral coil heat exchanger," *Int. J. Mech. Eng. Robot. Res.*, vol. 3, no. 2, pp. 370–375, Apr. 2014.