IJARSCT



International Journal of Advanced Research in Science, Communication and Technology

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

Volume 5, Issue 11, April 2025



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

Copyright to IJARSCT www.ijarsct.co.in



DOI: 10.48175/IJARSCT-25874



481