**International Workshop on Sustainable Energy, Power and Propulsion**

**January 19-21, 2024, IIT Delhi, India**

**Computational Fluid Dynamic analysis of fluid flow and heat transfer in modified triangular duct used in Solar air Heater**

|  |  |  |
| --- | --- | --- |
| Rajneesh Kumar Mechanical Engineering. DepartmentNational Institute of Technology, Hamirpur, India – 177005Email: Rajneesh127.nith@gmail.com | Dr. Varun GoelMechanical Engineering. DepartmentNational Institute of Technology, Hamirpur, India – 177005Email:varun7go@gmail.com | Prof. Anoop KumarMechanical Engineering. DepartmentNational Institute of Technology, Hamirpur, India – 177005Email: anoop@nith.ac.in |

Abstract

*A numerical study was carried out to investigate thermal and hydrodynamic characteristic of flow through modified equilateral triangular duct under steady state conditions. The triangular duct was modified by modifying (curved) sides of the duct with appropriate radius of curvature. The analysis was performed on three-dimensional model of the duct with hydraulic diameter of 0.044 m. The different values of radius of curvature were considered in this study. The walls of duct were subjected to constant heat flux boundary condition (1,000 Wm-2). The velocity of air flowing through duct was varied to achieve Reynolds number in the range of 4,000 to 15,000. The finite volume approach is used for discretization of flow governing equations (continuity, momentum and energy) with the help of ANSYS (Fluent) software. The flow equations were solved by using semi-implicit method for pressure linked equation (SIMPLE) method with pressure-velocity coupling. The different turbulence models available in Fluent were studied in order to choose best suitable for the modified duct. The renormalized (RNG)* $k-ε$ *turbulent model has been used for simulating heat transfer and fluid flow through duct. The complication of geometry was handled with multi-block meshing and hexahedral grids were generated in computational domain. The grids independence test was also performed to ensure the accuracy of results. The results were presented in terms of heat transfer coefficient and friction factor for the modified duct and compared with the simple duct under similar conditions. The curved wall has significant effect on heat transfer and fluid flow through the duct. The radius of curvature promotes fluid interactions in corner regions and helps in uniform heat transfer from the sides of the modified duct. With the modified sides, the heat transfer gets improved by 0.24 times the smooth duct.*

***Keywords*:** Modified Duct, Curved Sides, and Heat transfer and flow characteristics.