**Cfd analysis of solar absorber plate USING ansys fluent**

**Abstract**

Solar energy is becoming an alternative for the limited fossil fuel resources. One of the simplest and most direct applications of this energy is the conversion of solar radiation into heat, which can be used in water heating systems. A commonly used solar collector is the flat-plate. Solar flat plate collectors are used for domestic and industrial purposes and have the largest commercial application amongst the various solar collectors. This is mainly due to simple design as well as low maintenance cost. A lot of research has been conducted in order to analyze the absorber plate operation and improve its efficiency. An attempt is being made to analyze the solar absorber plate using the Computational Fluid Dynamics (CFD) so as to simulate the solar absorber plate for better understanding of the heat transfer capabilities of the absorber. In the present work, Fluid flow and heat transfer in the absorber panel are studied by means of Computational Fluid Dynamics (CFD). The conjugate heat transfer phenomenon between absorber and water is modeled using FLUENT CFD software. By using FLUENT CFD software solar heat absorber tube made of copper and aluminium are analyzed and compared to each other by passing different working fluids like water , air etc. The solar radiation heat transfer is not modeled; . The geometric model and fluid domain for CFD analysis is generated using ANSYS Design Modeler software, Grid generation is accomplished by ANSYS Meshing Software. The numerical results obtained using the CFD software for copper and aluminium for same inlet temperature has to be analyzed.