

Abstract

Mohamed Abd El Fatah Mohamed Teamah

Numerical Simulation of Turbulent Heat Transfer in Turbine Blades

Abstract: This paper presents a numerical simulation of turbulent heat transfer in turbine blades. The simulation is performed using a finite volume method (FVM) based computational fluid dynamics (CFD) software. The flow is modeled as a turbulent flow using a $k-\epsilon$ turbulence model. The geometry of the turbine blade is discretized using a structured mesh. The boundary conditions are specified as follows: inlet velocity of $U = 100 \text{ m/s}$, inlet temperature of $T = 500 \text{ K}$, and outlet pressure of $P = 1 \text{ atm}$. The wall temperature is assumed to be constant at $T_w = 1500 \text{ K}$. The simulation results show that the turbulent heat transfer coefficient is significantly higher than the laminar heat transfer coefficient. The maximum heat transfer coefficient is found at the leading edge of the blade, where it reaches a value of approximately $100,000 \text{ W/m}^2\text{K}$. The heat transfer coefficient decreases towards the trailing edge of the blade. The simulation also shows that the turbulent flow is characterized by a high degree of mixing, which leads to a more uniform temperature distribution across the blade surface. The results are compared with experimental data, and a good agreement is observed. The simulation results are presented in the form of contour plots and line graphs. The contour plots show the temperature distribution across the blade surface, and the line graphs show the variation of the heat transfer coefficient along the span of the blade. The simulation results are also compared with analytical solutions, and a good agreement is observed. The simulation results are used to optimize the design of turbine blades for improved performance and efficiency.