



Synopsis

Numerical Analysis of Air Film Cooling Effectiveness at High Pressure and Temperature

VEASNA MOM

Undergraduate Mechanical Engineering

Gujarat Technological University

The objective of this research migration project is to numerically simulate Air Film Cooling effectiveness on 2D Flat Plate Geometry using OpenFOAM-v2312. The simulation utilizes the **buoyantBoussinesqSimpleFoam** heat transfer solver for incompressible fluid to model the phenomenon on a 2D Flat Plate Geometry. The aim is to compare the results against the findings of the Li & Wang paper on Film Cooling, which was conducted using Ansys Fluent. The simulation investigates the effects of various parameters, including blowing ratio, mainstream temperature, and injection angle, on the cooling performance. The mainstream flow has a temperature of 400 K and a velocity of 10 m/s, while the coolant air is injected with a blowing ratio of 1.32, corresponding to velocity of 10 m/s, and a temperature of 300 K. All walls are considered adiabatic, with the top wall set to a slip condition and the remaining walls set to no-slip conditions. The turbulence dissipation rate and kinetic energy at the inlets are set to $\epsilon = 1 \text{ m}^2/\text{s}^3$ and $k = 1 \text{ m}^2/\text{s}^2$, respectively.

References

[1] Li, X., & Wang, T. (2006). Simulation of film cooling enhancement with mist injection.