Numerical Simulations of Condensing Moist Air Around Cold Cylinder

Adel E. Alshayji[1]

[1]Kuwait University, Kuwait

Abstract

In this study numerical simulations of moist air flow around cold cylinder are investigated. The condensation of moist air on the cold cylinder can effect the heat transfer and fluid flow around the cylinder. This physical phenomenon is encountered in a number of industrial processes such as in the oil and gas industry, food processing, thermal energy storage systems, and many others.
COMSOL is used in this study to simulate the flow around a round cylinder and focus on measuring the heat transfer and moister built up around the cylinder, by examining the effect of air speed, the environment relative humidity, and temperature. This problem is simplified to two-dimension, unsteady, and weakly compressible problem. The physics used in COMSOL for the study are selected to be laminar fluid flow, heat transfer in fluid with moist air, and transport of diluted species to track and control the concentration of moist air.
The study shows that thermal instability, due to natural convection effect especially when air speed is low, will affect the flow and heat transfer around the cylinder significantly. Studying flow over bodies is important because it models real life applications such as flow around buildings, chimneys, bridges, and heat exchanger pipes. Controlling this flow is essential especially in moist environment.

Figures used in the abstract

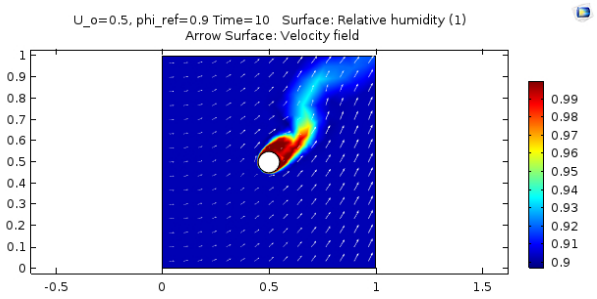

Figure 1: Relative humidity and air flow velocity around cold cylinder.