Numerical Investigation of Flow Past in a Staggered Tube Bundle

Authors : Kerkouri Abdelkadir

Abstract : Numerical calculations of turbulent flows are one of the most prominent modern interests in various engineering applications. Due to the difficulty of predicting, following up and studying this flow for computational fluid dynamic (CFD), in this paper, we simulated numerical study of a flow past in a staggered tube bundle, using CFD Code ANSYS FLUENT with several models of turbulence following: k-ε, k-ω and SST approaches. The flow is modeled based on the experimental studies. The predictions of mean velocities are in very good agreement with detailed LDA (Laser Doppler Anemometry) measurements performed in 8 stations along the depth of the array. The sizes of the recirculation zones behind the cylinders are also predicted. The simulations are conducted for Reynolds numbers of 12858. The Reynolds number is set to depend experimental results.

Keywords : flow, tube bundle, ANSYS Fluent, CFD, turbulence, LDA, RANS (k-ε, k-ω, SST)

Conference Title : ICESMA 2018 : International Conference on Energy Systems Modeling and Analytics

Conference Location : London, United Kingdom

Conference Dates : December 13-14, 2018