The phenomenon of the movement of fluid flow across an object (bluff body) plays an important role in engineering applications such as heat exchangers, combustion, and transportation. By varying the Reynolds number, different flow patterns and characteristics of the vortex shedding in the wake of a circular cylinder has been observed and discussed in the literature, but the wake characteristics of circular heated cylinder is more complicated to be analyzed because of the influence of physical buoyancy on the viscous phenomena occur. Based on the above ideas, then conducted research on the evaluation of a wide variety of turbulence models in the case of fluid flow through a heated cylinder.

This research was carried out numerically with the software Fluent 6.3.26 with unsteady solver models. Two different value of Reynolds Number, respectively - each Re = 135 and Re = 1000. The value of the Richardson Number is also set fixed at a value of 0 and 1 for Re = 135, while for Re = 1000, used the value Ri = 0 and Ri = 2.77. Later models used are Laminar viscous model and two turbulence models, namely k-ε realizble, and k-ω SST. Where is the fluid used is water on the boundary condition. Then the results of this study will be compared with the results of previous experiments that have been carried out.

From the research that has been done, it is known that there are differences in the results of the value of Strouhal number by varying the value of time step used during the iteration process. Where the smaller the value of time step used the more accurate the results obtained. And for the overall analysis, k-ω SST turbulence
models has the closest experimental results based on parameters were observed.

Keywords: Heated cylinder, turbulence models, unsteady flow