

Abstract

The research aims to analyze the work of cyclone against the coefficient of restitution. Research will use numerical study. The result of the CFD (Computational Fluid Dynamic) simulation will be compared with the result of experiments in other literature. Because the flow in the cyclone is believed to be included in the type of turbulence so that in CFD analysis it also needs to involve turbulent influences in solving the momentum equation. One turbulent models namely Reynolds Stress Model will be involved in numerical calculations. From the results of numerical calculations and experimental results it can be concluded that the cyclone design in this study still allows for a short cut flow. The amount of particle separation efficiency is above 90% and does not depend significantly on the velocity of gas entering.

Keywords : cyclone, COR, CFD, turbulence, Method

