CFD-Based Design Optimization and Experimental Validation of a Kitchen Hood Centrifugal Fan

Asawar Bin Riaz
Beko Corporate – Cooking Appliances Plant, Bolu, Turkey

Abstract

The centrifugal fan is the most crucial component of a domestic kitchen fume extractor, generating the required pressure gradient to capture cooking fumes and expel them through the exhaust duct. Apart from the impeller wheel and the motor, the volute design plays a critical role in determining the fluid efficiency and acoustic characteristics of the system. This study presents a computational fluid dynamics based optimization of a forward-curved centrifugal fan volute to improve airflow performance. ANSYS Fluent is used to perform steady-state three-dimensional simulations, where the fan is modelled using Multiple Reference Frame (MRF) approach. SST K-w was used for modelling turbulence kinetic energy, and a y+ of less than 10 was over the fan wheel. A validation study was conducted over the operating airflow range to assess the accuracy of the numerical model against experimental measurements. Based on the CFD analysis, an optimized volute geometry was developed and experimentally evaluated. The prototype demonstrated a 14% increase in airflow rate compared with the baseline design under same RPMs. A power savings of 48 Watt was realized at a 750 m3/h flowrate, a 27% improvement over the baseline. The CFD analysis helped identify regions of recirculation at exit of volute with the potential to improve noise characteristics. This study shows that CFD is an effective design tool for improving the aerodynamic performance of centrifugal fans used in domestic kitchen fume extractors.

Keywords

Computational Fluid Dynamics (CFD), Centrifugal Fan, Kitchen Fume Extractor.