Computational studies of flow through cross flow fans - Effect of blade geometry

Abstract

This present paper describes three dimensional computational analysis of complex internal flow in a cross flow fan. A commercial computational fluid dynamics (CFD) software code CFX was used for the computation. RNG k-ε two equation turbulence model was used to simulate the model with unstructured mesh. Sliding mesh interface was used at the interface between the rotating and stationary domains to capture the unsteady interactions. An accurate assessment of the present investigation is made by comparing various parameters with the available experimental data. Three impeller geometries with different blade angles and radius ratio are used in the present study. Maximum energy transfer through the impeller takes place in the region where the flow follows the blade curvature. Radial velocity is not uniform through blade channels. Some blades work in turbine mode at very low flow coefficients. Static pressure is always negative in and around the impeller region.