Proceedings of the 9th World Congress on Momentum, Heat and Mass Transfer (MHMT'24)
London, United kingdom – April 11 – 13, 2024
Paper No. ENFHT 294
DOI: 10.11159/enfht24.294

Flow Analysis of an Ultrathin Forward-Curved Multi-Blade Centrifugal Fan at High Revolutionary Speed

Weiwei Liao[1], Yijun Mao[2], Xiaobing Luo[1*]
[1]School of Eenergy and Power Engineering, Huazhong University of Science and Technology, Wuhan 430074, China
[2]School of Aerospace Engineering, Huazhong University of Science and Technology, Wuhan 430074, China
*Corresponding author Xiaobing Luo, E-mail: luoxb@hust.edu.cn

Extended Abstract

Ultrathin centrifugal fan is widely used in small-scale electronics for air forced cooling which is characterized by large inverse pressure gradient and small axial dimension. These two features will induce flow separation at the blade tip and boundary layer effect amplification, which make the flow field of ultrathin centrifugal fan more complicated. Therefore, it's worth investigation.

Concentrating on the two main characteristics mentioned above, flow field analysis on ultrathin centrifugal fan at 5124 rpm is conducted based on the LES method, which uses the WALE sub-grid model to simulate the sub-grid scale turbulence effect. To make sure the temporal and spatial resolution is adequate for the LES, courant number is set to 1 and resolved turbulent kinetic energy makes up 80 precents of the total turbulent energy[1].

Firstly, to make sure the simulation accuracy is adequate, the hot-wire thermometer measurement is conducted. A total amount of 25 monitoring points in line are set at the outlet of the volute. The turbulence length scale and time scale and RMS velocity are calculated. The RMS velocity reaches its maximum near the centre of outlet region and the timescale's trend is different and reaches its maximum near the volute tongue. As timescale characterize the time spent for the large eddy traverse the inertial range to the dissipation regime [2], this suggests the turbulence length scale near the volute tongue is large there which needs more time to dissipate. And the turbulence spectrum is obtained to compare with the simulation spectrum, the results indicating the simulation and experiment results align well. The turbulence spectrum contain 2 regions where -5/3 power law shows in frequency range 1000-5000 Hz and -5 power law shows in frequency range above 5000Hz indicating that the turbulence intensity is very high.

After steady and transient simulation, the boundary layer thickness is calculated to account for 10% of the total axial height of the volute, which is significant and nonnegligible for the air flow and will induce flow blockage. In terms of the asymmetry of the blade and inlet configuration, the boundary layer also exhibits asymmetric velocity profile to some extent. Near the hub plane, the velocity profile is more fluctuated. Near the volute tongue, the boundary layer is thickened which will cause flow blockage undoubtedly.

The flow separation feature is investigated by Q-criterion analysis[3] and entropy generation analysis. Through analysis, the flow separation presents circumferential asymmetry, where the blade tip of the pressure side near the outlet region of the volute is prone to induce flow separation. Moreover, the vortex presents non-uniformity in the height dimension. Four cutting planes are inserted to investigate the vorticity distribution and two valleys of q magnitude are observed in flow passage indicating two vortex cores exist in one single flow passage. And the wall shear stress distribution on the blade surface confirms that point.

Through the entropy generation analysis, entropy production rate can be categorized into 3 types[4]: time averaged and fluctuating terms and wall entropy production term. After calculation, the time averaged term of entropy production rate in the volute region reaches $1.66e^{-4}$ W/K, making up 49.4% of the total entropy production. The fluctuating term of entropy generation in rotor is greater than that in the volute region because unstable turbulence is more intense in rotating region. According to the contour of entropy production rate, we can find the blade wake region near the outlet is the main area where entropy production rate is high, corresponding to the area where q criterion differs a lot.

The internal flow field of an ultrathin centrifugal fan is analysed. This work will provide insights on how to improve the efficiency of an ultrathin centrifugal fan.

References

[1] E. Sorguven, S. Incir, and J. Highgate, 'Understanding loss generation mechanisms in a centrifugal pump using large eddy simulation', *International Journal of Heat and Fluid Flow*, vol. 96, p. 108994, Aug. 2022, doi: 10.1016/j.ijheatfluidflow.2022.108994.

[2] S. B. Pope, Turbulent Flows. Cambridge: Cambridge University Press, 2000.

[3] Y. Zhang, K. Liu, H. Xian, and X. Du, 'A review of methods for vortex identification in hydroturbines', *Renewable and Sustainable Energy Reviews*, vol. 81, pp. 1269–1285, Jan. 2018, doi: 10.1016/j.rser.2017.05.058.

[4] L. Zhou *et al.*, 'Application of entropy production theory for energy losses and other investigation in pumps and turbines: A review', *Applied Energy*, vol. 318, p. 119211, Jul. 2022, doi: 10.1016/j.apenergy.2022.119211.