Feedback Studio - Microsoft Edge
https://ev.turnitin.com/app/carta/en_us/?s=1&o=1228347463&u=1083371239&lang=en_us
turnitin
ARPN
18 of 20

VOL. 10, NO. 16, SEPTEMBER 2015 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences
©2006-2015 Asian Research Publishing Network (ARPN). All rights reserved.
www.arpnjournals.com

TURBULENT FLOW ANALYSIS IN AUXILIARY CROSS-FLOW RUNNER OF A PROTO X-3 BIOENERGY MICRO GAS TURBINE USING RNG k-ε TURBULENCE MODEL

Steven Darmawan[1,2], Ahmad Indra Siswantara[2], Budiarso[3], Asyari Daryus[2,3], Agus Tri Gunawan[2], Achmad Bayu Wijayanto[3] und Harto Tanujaya[2]
[1]Department of Mechanical Engineering, Universitas Indonesia, Depok, Jawa Barat, Indonesia
[2]Department of Mechanical Engineering, Universitas Tarumanagara Jl. Let.Jen S. Parman No.1 Jakarta, Indonesia
[3]Department of Mechanical Engineering, Universitas Darma Persada, Jl. Radin Inten II, Pondok Kelapa, Jakarta Indonesia
E-Mail: stevend@ft.untar.ac.id

ABSTRACT

Simple and wide range application of cross-flow runner has lead its application to a Proto X-3 Bioenergy Micro Gas Turbine (MGT) that has been developed. The MGT is a dual-stage radial compressor-turbine type. Furthermore, highly turbulent flow inside the cross-flow fan need accurate analysis. CFD method with RNG k-ε choosen based on models characteristics. This paper analyzed the flow inside the fan based on the experimental data of the MGT and the result represented by several parameters of turbulent flow. The simulation condition were assumed to be isothermal due to the small temperature difference with the turbulent Prandtl number $Pr_t = 1$. The result shows several specific vortex inside the runner. Recirculation flow ethat caused the eccentric vortex occurs at the inner side while the throughflow occurs at the outerside of the fan. The mass flow ratio conducted by CFD simulation shows a good agreement with the actual mass flow rate of the cross-flow runner. The results that presented by velocity magnitude, absolute pressure, eddy dissipation, and turbulence kinetic energy shows a realistic on each turbulent parameter based on they treads. This results shows that the method used is prospective to be applied both on analysis or design of the cross-flow runner.

Keywords: cross-flow runner, MGT, turbulent flow, RNG k-ε turbulence model, recirculation vortex.

INTRODUCTION

Cross-flow runner (CFR, hereafter) is a atmospheric radial turbine that generates power by converting kinetic energy to mechanical energy which based on Banki turbine (cross-flow turbine) [1]. CFR system consist of two main components; the runner itself and the nozzle with square cross-section. CFR design based on three main characteristics; simple construction, low-cost, and maximum efficiency and has been prospected as Renewable Energy Resources (RES) for under 3MW hydroelectic generator system [2], [3]. This characteristics has lead the applications of CFR as power extractor of a Proto X-3 Bioenergy Micro Gas Turbine (MGT) prototype that has been developed which is a dual-stage compressor-turbine [4]. The CFR was driven by the inlet air of the first stage compressor. The high flow air to first stage made it possible to drive the cross-flow fan. Basically, MGT is developed for small power generation upto 200 kW [5], [6]. Some of the advantages of this MGT are high power to weight ratio, high tolerance to many kind of liquid and gaseous fuels and biofuels has made this prime mover suitable to be used in Zero-Energy-Building-based [7-10].

During the operation, as a turbomachine, the suction and discharge of CFR occurs radially, generated highly turbulent flow for recirculating and reserve flow [11]. Many experimental and numerical analysis shows the flow complexity of CFR. Experimental study by numbers of researcher has lead to description of two main vortex inside the CFR; throughflow at the outside region and recirculation flow at the inside region [12-17].

Handling and costly experiment as well as the accomodation has made numerical method also conducted on last decade with many turbulence model. Kaniecki use RNG k-ε found the throughflow as well as the recirculation flow [18]. Cheng use STD k-ε in numerical analysis the flow of CFR [19]. RNG k-ε also uses by Toffolo (2005) [16]. Hirata et.al in 2008 use STD k-ε [20]. The current numerical analysis by Sun et.al also *RNG k-ε* turbulence model for more detailed flow [17]. These numerical results show there are two main vortexes occur in CFR; the eccentric vortex at recirculation region and vortex in the throughflow region. Since the flow inside the CFR is undoubtly turbulent, flow analysis of turbulent flow with suitable turbulence model is needed to analysis and can be used for future development of the CFR system. Despite the STD k-ε as the most widely used turbulence model, this model also reported since that model is overpredicts the dissipation rate [21-24]. RNG k-ε turbulence model developed by Yakhot & Orszag [25] for recirculating flow has became alternatives to predict such flows [26], [27].

The aim of this paper is to analyze numerically the flow inside the cross-flow runner with RNG k-ε turbulence model since this model is developed to such flow. The turbulent momentum diffusivity and turbulent thermal diffusivity is also assumed to be equal ($Pr_t = 1$) and the deafult constant to the Yakhot & Orszag model was used. Optimal flow characteristics of cross-flow runner on Proto X-3 can be used to optimized the system to be a compact prime mover system.

Match Overview
15%
1 Submitted to Internatio... 4% Student Paper
2 Budiarso, , Ahmad Indr... 4% Publication
3 Siswantara, Ahmad Ind... 4% Publication
4 www.scientific.net 3% Internet Source
Page: 1 of 6 Word Count: 3544 Text-only Report High Resolution On