Powder Technology 364 (2020) 152–158

Contents lists available at ScienceDirect

Powder Technology

journal homepage: www.elsevier.com/locate/powtec

Influence of divergent length on the gas-particle flow in dual hose dry ice blasting nozzle geometry

Mohamad Nur Hidayat Mat [a], Nor Zelawati Asmuin [a], Md Faisal Md Basir [b], Marjan Goodarzi [c,*], Muhammad Faqhrurrazi Abd Rahman [a], Riyadhthusollehan Khairulfuaad [a], Balasem Abdulameer Jabbar [d], Mohd Shareduwan Mohd Kasihmuddin [e]

[a] Department of Energy and Thermodynamic Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn, 86400 Parit Raja, Johor, Malaysia
[b] Department of Mathematical Sciences, Faculty of Science, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia
[c] Sustainable Management of Natural Resources and Environment Research Group, Faculty of Environment and Labour Safety, Ton Duc Thang University, Ho Chi Minh City, Vietnam
[d] Engineering Technical College – Najaf, AL- Furat AL-Awsat Technical University, Najaf, Iraq
[e] School of Mathematical Sciences, Universiti Sains Malaysia, 11800 USM Penang, Malaysia

ARTICLE INFO

Article history:
Received 31 July 2019
Received in revised form 1 December 2019
Accepted 22 January 2020
Available online 23 January 2020

Keywords:
Divergent length
Dry ice blasting
CFD
Nozzle geometry
Particle gas flow
Simulation

ABSTRACT

A numerical simulation study was performed to examine the effect of nozzle geometry and divergent length on gas-particle flows in dual hose dry ice blasting. The simultaneous model of mass momentum and energy exchange between two phases was solved iteratively. The phases are solid dry ice particle and compressible air fluid as a working medium. The results were presented in the gas flow field along the nozzle centerline. The results showed that increasing divergent length decreases the gas-particle density along the centerline; and the density development along the nozzle centerline decreased along the length. Eddy viscosity of the nozzle cavity was steady for length of 0.25 m, but increased drastically for higher values. The substantial increase in eddy viscosity after that position is due to the mixing flow between dry ice pellet and compressed gas which is diverging out from the nozzle outlet area. Thus, the mixing flow experienced back pressure from the atmospheric pressure. The particle mass concentration decreased by increasing nozzle centerline coordinates. The pressure increased along the length; however, it dropped at the end due to the reverse pressure. The temperature increased steadily because of conversion of turbulence to internal thermal energy. The characteristics of gas-particle flow in the nozzle cavity provide better understanding of multiphase flow in turbine and jet engine flow analysis.

1. Introduction

Dry ice blasting does little harm to the environment. It is a form of carbon dioxide (CO_2) cleaning process in which dry iceor the solid form of CO_2 is accelerated in a pressurized air stream and directed toward contaminated surfaces. The cleaning process using dry ice blasting can be categorized into three active mechanisms which are thermal, mechanical and sublimation effect (Fig. 1).

Mechanisms of surface removal are described as follows:

- The thermal effect happens due to difference in the thermal coefficient between surface contaminant and substrate, which causes brittle detachment of the contaminant.
- Mechanical effects are derived from the kinetic energy of the blasting, which is capable of achieving supersonic speed upon the surface impact.
- The sublimation effect occurs due to the ability of CO_2 to change from solid to a gaseous phase, resulting in 800 times volume increase [1]

In terms of primary advantages, dry ice blasting is a relatively low-abrasion blasting. In addition, it is environmentally friendly because the blast media immediately disappears upon impact and returns to its natural state in the atmosphere. Therefore, it produces little waste other than the contaminant being removed. In addition, the carbon dioxide released from the dry ice blasting process is not a new carbon dioxide. Thus, it does not increase CO_2 concentration in the atmosphere.

Nozzle geometry plays an important role in optimizing the cleaning performance of surface treatment [2,3]. The optimum

* Corresponding author.
E-mail address: marjan.goodarzi@tdtu.edu.vn (M. Goodarzi).

https://doi.org/10.1016/j.powtec.2020.01.060
0032-5910/

© Universiti Tun Hussein Onn Malaysia Publisher's Office

IJIE

http://penerbit.uthm.edu.my/ojs/index.php/ijie
ISSN : 2229-838X e-ISSN : 2600-7916

The International Journal of Integrated Engineering

CFD Investigation of Empty Flanged Diffuser Augmented Wind Turbine

Balasem Abdulameer Jabbar Al-Quraishi[1,2*], Nor Zelawati Asmuin[1], Nurul Fitriah Nasir[1], Noradila Abdul Latif[1], Juntakan Taweekun[3], Sofian Mohd[1], Akmal Nizam Mohammed[1], Wisam A. Abd Al-Wahid[2]

[1]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia (UTHM), 86400 Parit Raja, Batu Pahat, Johor, MALAYSIA

[2]Engineering Technical College - Najaf, AL-Furat Al-Awsat Technical University, Najaf, IRAQ

[3]Department of Mechanical Engineering, Faculty of Engineering,
Prince of Songkla University, 15 Karnjanavanich Road, Hat Yai, Songkhla 90110, THAILAND

*Corresponding Author

DOI: https://doi.org/10.30880/ijie.2020.12.03.004
Received 20 May 2018; Accepted 20 September 2018; Available online 27 February 2020

Abstract: Enclosing a wind turbine within a flanged diffuser is an innovative mean to increase the powerharvested by turbine blades and it is among the most effective devices for increasing wind turbine energy. The geometric parameters of the empty flanged diffuser contribute efficiently to increase mass flow in the diffuser, hence improve the turbine performance. The study presents developed models of the geometrical parameters of an empty flanged diffuser that suitable for a scaled-down (1-6.5) horizontal axis wind turbine, the geometry parameters were involved the diffuser length, diffuser angle, flange height and flange angle. The geometrical models were verified and CFD investigated in 2-D and 3-D domains. Results obtained from CFD simulations show that, using a compact size of flanged diffuser within optimum geometrical parameters can give well acceptable for flow velocity increase at suggested place for the turbine rotor install. The increase in flow velocity is due to lower pressure at the outlet of the diffuser. As there is also a significant effect of the flange angle on increasing the flow velocity inside the diffuser where the rate of increase in wind velocity at turbine position was calculated for two flange angles (0 ˚ and 5 ˚). In another hand, the results also provided information on the velocity contours and velocity streamlines around diffuser geometry.

Keywords: Wind energy harvesting, DAWT, flanged diffuser.

1. Introduction

The need for energy to consume society increases as technology advances in certain areas, so the capability to produce energy must keep pace with increasing demands. Due to the rapid depletion of fossil energy sources, there is a necessary need to seek alternative and sustainable sources of energy. However, wind energy as a renewable and inexhaustible source of energy is now the fastest-growing energy technology worldwide (T. Wei, 2010). Wind power systems, represented by wind turbines have been the focus of interest of scientists and researchers in the past decades. Flowing of wind through the turbine rotor leads to the production of mechanical energy that can be used in many applications especially to produce electricity.

However, power produced by the wind turbine is dependent on the Betz limit; an ideal type can extract only 59.3% of incoming energy in stream-tube by turbine blades (Libii & Drahozal, 2012), (IGRA, 1981). As the energy extracted

*Corresponding author: balasemalquraishi@atu.edu.iq; balasemalquraishi@gmail.com

التحقيق في توربين الرياح المعززذي الحافة باستخدام ديناميكا الموائع الحسابية (CFD)

الخلاصة:

يعد تضمين توربين رياح داخل ناشر ذو حواف وسيلة مبتكرة لزيادة الطاقة التي يتم حصادها بواسطة شفرات التوربين وهي من بين أكثر الأجهزة فعالية لزيادة طاقة توربينات الرياح. تساهم المعلمات الهندسية للناشر ذو الحواف الفارغة بكفاءة في زيادة تدفق الكتلة في الناشر ، وبالتالي تحسين أداء التوربين. تقدم الدراسة نماذج متطورة للمعلمات الهندسية لناشر ذو شفة فارغة مناسبة لتوربين رياح المحور الأفقي المصغر (1-6.5) ، وقد اشتملت المعلمات الهندسية على طول الناشر وزاوية الناشر وارتفاع الشفة وزاوية الحافة. تم التحقق من النماذج الهندسية والتحقيق في CFD في المجالات ثنائية الأبعاد وثلاثية الأبعاد. تظهر النتائج التي تم الحصول عليها من محاكاة CFD أنه باستخدام حجم مضغوط من الناشر ذو الحواف ضمن المعلمات الهندسية المثلى يمكن أن يعطي مقبولًا جيدًا لزيادة سرعة التدفق في المكان المقترح لتركيب دوار التوربين. ترجع الزيادة في سرعة التدفق إلى انخفاض الضغط عند مخرج الناشر. كما أن هناك أيضًا تأثيرًا كبيرًا لزاوية الحافة على زيادة سرعة التدفق داخل الناشر حيث تم حساب معدل الزيادة في سرعة الرياح عند موضع التوربين لزاويتين شفة (0° و 5)°. من ناحية أخرى ، قدمت النتائج أيضًا معلومات حول ملامح السرعة وتبسيط السرعة حول هندسة الناشر.