Energy 148 (2018) 1103–1115

Contents lists available at ScienceDirect

Energy

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

Review

A comparative analysis on the uniformity enhancement methods of solar thermal drying

Shaymaa Husham Abdulmalek [a], Morteza Khalaji Assadi [a, *], Hussain H. Al-Kayiem [a], Ali Ahmed Gitan [b]

[a] Universiti Teknologi PETRONAS, Faculty of Engineering, Mechanical Engineering Department, Solar Thermal Advanced Research Center, Bandar Seri Iskandar, Perak, Malaysia
[b] Tikrit University, Faculty of Engineering, Mechanical Engineering Department, Tikrit, Iraq

ARTICLE INFO

Article history:
Received 2 November 2017
Received in revised form
13 December 2017
Accepted 11 January 2018
Available online 3 February 2018

Keywords:
Hybrid solar thermal energy
Drying
Desiccant
Uniform drying

ABSTRACT

The uniformity of solar drying process and the quality of the product are inter-related parameters. Drying uniformity is influenced by the significant process air properties which are temperature, humidity and velocity. Accordingly, solar drying uniformity may be improved by integration with dehumidification system and/or optimizing the dryer design. These concepts were reviewed extensively in this paper by brush up the solar thermal hybrid dryers, the performance of solar assisted desiccant systems for dehumidification of drying air, the effect of geometrical parameters on drying performance, and the drying performance of different products. In the context of desiccant systems, the performance of drying is influenced by desiccant material, dehumidifier design and regeneration technique used. While, the issue of solar dryer design is related to drying chamber geometrical parameters, considering multiple drying chambers, and modeling and optimization of dryer design. Coming out with this comprehensive review may motivate to enhance the quality of product and drying performance in terms of cost and time.

Contents

* Corresponding author.
E-mail addresses: shaymaa_g03363@utp.edu.my (S. Husham Abdulmalek), morteza.assadi@utp.edu.my (M. Khalaji Assadi), hussain_kayiem@utp.edu.my (H.H. Al-Kayiem), aliagitan@tu.edy.iq, aliahmedeng1@gmail.com (A.A. Gitan).

https://doi.org/10.1016/j.energy.2018.01.060
0360-5442/

ISSN: 1813-162X (Print) ; 2312-7589 (Online)

Tikrit Journal of Engineering Sciences

available online at: http://www.tj-es.com

TJES Tikrit Journal of Engineering Sciences

Ataalah Hussain Jassim [1]

M.M. Rahman [2]

Khalaf Ibrahim Hamada [1]

M. Ishak [2]

Tahseen Ahmad Tahseen [1,*]

[1] Department of Mechanical Engineering
College of Engineering
Tikrit University
Tikrit, Iraq

[2] Faculty of Mechanical Engineering
Universiti Malaysia Pahang
26600 Pekan
Pahang, Malaysia

***Keywords*:**

In-line flat tube
finite volume technique
CFD
modeling
friction factor
artificial neural networks

ARTICLE INFO

Article history:

Received	14 November 2017
Accepted	06 March 2018
Available online	17 June 2018

Hybrid CFD-ANN Scheme for Air Flow and Heat Transfer Across In-Line Flat Tubes Array

ABSTRACT

Flat tubes are vital components of various technical applications including modern heat exchangers, thermal power plants, and automotive radiators. This paper presents the hybridization of computational fluid dynamic (CFD) and artificial neural network (ANN) approach to predict the thermal-hydraulic characteristics of in-line flat tubes heat exchangers. A 2D steady state and an incompressible laminar flow in a tube configuration are considered for numerical analysis. Finite volume technique and body-fitted coordinate system are used to solve the Navier–Stokes and energy equations. The Reynolds number based on outer hydraulic diameter varies between 10 and 320. Heat transfer coefficient and friction are analyzed for various tube configurations including transverse and longitudinal pitches. The numerical results from CFD analysis are used in the training and testing of the ANN for predicting thermal characteristics and friction factors. The predicted results revealed a satisfactory performance, with the mean relative error ranging from 0.39% to 5.57%, the root-mean-square error ranging from 0.00367 to 0.219, and the correlation coefficient (R^2) ranging from 99.505% to 99.947%. Thus, this study verifies the effectiveness of using ANN in predicting the performance of thermal-hydraulic systems in engineering applications such as heat transfer modeling and fluid flow in tube bank heat exchangers.

DOI: http://dx.doi.org/10.25130/tjes.25.2.08

نظام ديناميكا الموائع الحسابية والشبكة العصبية الاصطناعية الهجين لانتقال الحرارة وجريان الهواء عبر حزمة انابيب مسطحة مرتبة بشكل خطي

الخلاصة

الأنابيب المسطحة هي المكونات الحيوية لمختلف التطبيقات التقنية بما في ذلك المبادلات الحرارية الحديثة، ومحطات الطاقة الحرارية، ومشعات السيارات. يعرض هذا البحث استخدم عملية التهجين لديناميك الموائع الحسابية (CFD) والشبكة العصبية الاصطناعية (ANN) للتنبؤ بالخصائص الحرارية والهيدروليكية للمبادلات الحرارية ذات الانابيب المسطحة. اعتبرت حالة الجريان مستقرة وبعدين وطباقي التدفق وغير قابل للانضغاط في التحليل العددي. استخدمت تقنية الحجم المحدود (finite volume technique) ونظام تطابق الاحداثيات (body-fitted coordinate) لحل معادلات نافير- ستوك (Navier–Stokes) ومعادلة الطاقة. حسب عدد رينولدز على أساس القطر الهيدروليكي الخارجي للأنبوب وبمدى 10 الى 320. تم تحليل معامل انتقال الحرارة والاحتكاك لجميع الحالات المدروسة للمسافة بين مركز الانابيب العرضية والطولية. استخدمت النتائج العددية من تحليل الـ (CFD) في تدريب واختبار الشبكة العصبية الاصطناعية للتنبؤ بالخصائص الحرارية وعامل الاحتكاك. أظهرت النتائج المتوقعة من الشبكة العصبية الاصطناعية أداء مرضيا، حيث تراوح متوسط الخطأ النسبي بين 0.39٪ و5.57٪، وكان مدى مربع متوسط جذر الخطأ من 0.00367 إلى 0.219، وتراوح معامل الارتباط (R^2) من 99.505٪ إلى 99.947٪. وبالتالي، فإن هذه الدراسة تتحقق من فعالية استخدام الشبكة العصبية الاصطناعية في التنبؤ في أداء المنظومات الحرارية والهيدروليكية وكذلك في التطبيقات الهندسية المختلفة مثل نمذجة انتقال الحرارة وتدفق الموائع في المبادلات الحرارية ذات حزم الانابيب.

1. INTRODUCTION

The fluid flow and heat transfer in tube banks demonstrate the real-life applications of various industrially significant processes. Tube bundles are widely employed in cross-flow heat exchangers, and their design is based on the empirical correlations of heat transfer and pressure drop. Cross-flow heat exchangers with tube banks are essential to numerous thermal and chemical engineering processes [1–4]. Flat tube designs have been recently introduced for modern heat exchanger applications such as automotive radiators. Unlike circular

* *Corresponding author: E-mail* : tahseen@tu.edu.iq ; tahseen444@gmail.com

Isı Bilimi ve Tekniği Dergisi, 38, 1, 55-64, 2018
J. of Thermal Science and Technology
©2018 TIBTD Printed in Turkey
ISSN 1300-3615

NUMERICAL INVESTIGATION OF THE FLAME LOCATION OF TURBULENT PREMIXED COMBUSTION IN A DIFFUSER BURNER EXPOSED TO VARIOUS TURBULENCE INTENSITIES AND TURBULENCE LENGTH SCALES

Ibrahim Thamer NAZZAL*, and Özgür ERTUNÇ***
* Mechanical Engineering Department, Özyeğin University, Çekmeköy, Istanbul 34794, Turkey
thamer.nazzal@ozu.edu.tr, ozgur.ertunc@ozyegin.edu.tr
** Mechanical Engineering Department, College of Engineering, Tikrit University, Tikrit, Iraq

(Geliş Tarihi: 20.03.2017, Kabul Tarihi: 15.12.2017)

Abstract: This study aims to investigate the response of the flame location of a turbulent premixed flame that has been exposed to various turbulence intensities and turbulence length scales. A diffuser-type burner is used to reveal the influence of turbulence intensity and turbulence length scales on the flame location of premixed propane–air flames without changing the inlet velocity of the fuel. Numerical simulations are performed for the turbulent premixed propane flames by using a coherent flame model under steady-state conditions. Results show that the flame location moves toward the inlet of the diffuser combustor with an increase in turbulence intensity for moderate and high turbulence length scales. The behavior of the flame location is different for the low turbulence length scale. The flame location initially decreases with an increase in turbulence intensity and subsequently stabilizes. Furthermore, the maximum flame area density is shown to increase with an increase in the turbulence intensity and the turbulence length scale, as the flame moves toward the inlet in these cases. It is clearly documented how turbulence intensity and turbulence length scale simultaneously influence the flame area density, flame shape, and flame location in a diffuser-type burner.
Keywords: Premixed turbulent combustion, flame area density, turbulence intensity, turbulence length scale, coherent flame model.

DİFÜZÖR TİPİ YANMA ODASINDA GERÇEKLEŞEN ÖN KARIŞIMLI TÜRBÜLANSLI YANMADA ORTAYA ÇIKAN ALEVİN KONUMUNUN TÜRBÜLANS YOĞUNLUĞU VE TÜRBÜLANS UZUNLUK ÖLÇÜSÜ İLE DEĞİŞİMİNİN SAYISAL OLARAK İNCELENMESİ

Özet: Bu makalenin amacı, ön karışımlı yanma sonucu oluşan alevin, çeşitli türbülans yoğunluklarına ve türbülans uzunluk ölçeğine maruz kalması sonucu oluşan alev yeri değişikliğini incelemektir. Yakıtın yanma odasına giriş hızını değiştirmeden, sadece türbülans yoğunluğunun ve türbülans uzunluk ölçeğinin alev yeri üstüne etkisini görebilmek için araştırmalar difüzör tipi yanma odasında gerçekleştirilmiştir. Propanın türbülanslı ön karışımlı yanma simulasyonları tutarlı alev modeli (coherent flame model) kullanılarak kararlı akış rejiminde gerçekleştirilmiştir. Orta ve yüksek türbülans uzunluk ölçeği kullanıldığında türbülans yoğunluğundaki artış ile alevin difüzörün girişine doğru hareket ettiği gözlemlenmiştir. Düşük uzunluk ölçeği kullanılarak gerçekleştirilen simulasyonlarda, alev türbülans yoğunluğunun artması ile girişe doğru yaklaştığı ancak türbülans yoğunluğunun daha da arttılmasına rağmen alev konumunda kayda değer bir değişiklik olmadığı gözlemlenmiştir. Sonuçlar, türbülans yoğunluğu ve türbülans uzunluk ölçeğindeki artışın, maksimum alev alan yoğunluğunu arttırdığını göstermektedir. Dahası türbülans yoğunluğunun ve uzunluk ölçeğinin alev alan yoğunluğu, alevin şekli ve konumu üzerinde aynı anda etkili olduğu gösterilmiştir.
Anahtar Kelimler : Türbülanslı ön karışımlı yanma, alev alan yoğunluğu, türbülans yoğunluğu, türbülans uzunluk ölçeği, tutarlı alev modeli.

NOMENCLATURE

A	Area [m^2]	FL	Flame location [m]
b	Progress reaction variable	FAD_{max}	Maximum flame area density [m2/kg]
C_p	Specific heat [kJ/kg-K]	IFA	integrated flame area [m^2]
D	Diameter [m]	Ka	Karlovitz number [chemical time scale / time
D_t	Mass diffusivity [m^2/s]	time]	

Int. J. Exergy, Vol. 27, No. 3, 2018 393

Static and dynamic analyses for the exergetic, exergoeconomic and environmental assessment of a high-performance building

Raaid Rashad Jassem Al Doury*

Department of Mechanical Engineering,
Özyeğin University,
Istanbul 34794, Turkey
Email: raaid.aldoury@ozu.edu.tr
and
Department of Mechanical Engineering,
Engineering College,
University of Tikrit,
Salah Al-Deen, Iraq
Email: raaidaldoury@tu.edu.iq
*Corresponding author

M. Pinar Mengüç

Center for Energy, Environment and Economy (CEEE),
Department of Mechanical Engineering,
Özyeğin University,
Istanbul 34794, Turkey
Email: pinar.menguc@ozyegin.edu.tr

Abstract: This paper presents an exergetic, exergoeconomic and environmental assessment of an existing high-performance building using both static and dynamic analyses. The IEA ECBCS framework is adopted for exergy analysis, whereas the SPECO method is used to implement the exergoeconomic analysis. The environmental impact is considered in a quantitative fashion. It is noted that a dynamic analysis using average hourly temperatures is preferred over a static analysis. However, if a simpler static analysis is to be used, an annual average temperature needs to be identified for specific climate zone and building type. For Istanbul, an average temperature of 14°C is recommended.

Keywords: exergy; exergoeconomic; environmental; static analysis; dynamic analysis; low-energy buildings; high-performance buildings.

Reference to this paper should be made as follows: Al Doury, R.R.J. and Mengüç, M.P. (2018) 'Static and dynamic analyses for the exergetic, exergoeconomic and environmental assessment of a high-performance building', *Int. J. Exergy*, Vol. 27, No. 3, pp.393–418.

Biographical notes: Raaid Rashad Jassem Al Doury is an Associate Professor of Mechanical Engineering at the University of Tikrit, Salah Al-Deen, Iraq. He received his BSc and MSc from the University of Tikrit in 1998 and 2001, respectively. His recent publications include 'Effect the form of perforation on the heat transfer in the perforated fins', in *Academic Research International*,

ISSN: 1813-162X (Print) ; 2312-7589 (Online)

Tikrit Journal of Engineering Sciences

available online at: http://www.tj-es.com

TJES Tikrit Journal of Engineering Sciences

Viscosity Effect on Stiffness of Non-conventional (Five Tilted Pads) Journal Bearing

Mohammed Oawed Atteaa Alhassany

Ali Khalid Aldulaimy *

Department of Mechanical Engineering
College of Engineering
Tikrit University
Tikrit
Iraq

***Keywords*:**

Tilted pad
stiffness of journal bearing
viscosity effect
tribology

ARTICLE INFO

Article history:
Received 12 March 2018
Accepted 15 April 2018
Available online 01 September 2018

ABSTRACT

In this tribological study, we highlight the effect of lubricating oil viscosity in the Multi-pads hydrodynamic journal bearings generate important improvement in characteristics of stiffness and stability in the high speed turbomachines. Depending on viscosity of oil film (three values) variation for five tilted pads bearing, each pad is pivoted and is facilitated to be tilted with small angles, by using Matlab program, we calculate the oil film thickness for convergence layer. We applied Reynold's equation and solved it's numerically by using finite difference method with 5 nodes technique to find the pressure distributed on each node in the mesh of tilted pad, then calculate stiffness coefficients. Results show that there is clear effect on stiffens with viscosity change. The increase in value of *Krr* (for $n = 0.3$) between viscosity (0.04 Pas. s) and viscosity (0.058 Pas. s) is14.33 MN/m, while the increase in *Krr* value between viscosity (0.058 Pas. s) and viscosity (0.087 Pas. s) is 11.37 MN/m. the increase in value the of *Kss* (for $n = 0.3$) between viscosity (0.04 Pas. s) and viscosity (0.058 Pas. s) is5.921 MN/m, while increase in Kss value between viscosity (0.058 Pas. s) and viscosity (0.087 Pas. s) is9.55 MN/m respectively. the increase in value of *Ksr* (for n = 0.3) between viscosity (0.04 Pas. s) and viscosity (0.058 Pas. s) is 8.95 MN/m, while the increase in *Ksr* value between viscosity (0.058 Pas. s) and viscosity (0.087 Pas. s) is 14.41 MN/m respectively. the increase in value of *Krs* (for $n = 0.3$) between viscosity (0.04 Pas. s) and viscosity (0.058 Pas. s) are 5.08 MN/m, while the increase in *Krs* value between viscosity (0.058 Pas. s) and viscosity (0.087 Pas. s) is8.19 MN/m respectively. The values of the dominate principal coefficients *Krr* is greater than that of *Ksr*, also The values of the principal coefficients *Kss* is greater than that of cross coupling *Krs* for all values of viscosity that studied. From this result, we can conclude the side effect of cross coupling coefficients (*Ksr* ,*Krs*) can be overcome by great values for principal coefficient (Krr, Kss) respectively, so we can get good improvement instability for this bearing by variation the viscosity. After that, we regarded to use high viscosity lubricant in multi-pad journal bearing to improve the performance and stability by controlling the stiffness coefficients.

DOI: http://dx.doi.org/10.25130/tjes.25.3.09

تأثير لزوجة الزيت على معامل نابضية المساند الغير تقليدية (ذو خمس وسادات قابلة للإمالة)

الخلاصة

تم في هذه الدراسة التراييولوجية البحث في تأثير لزوجة المزيت المستخدم في المساند الهيدروديناميكية ذات الخمس وسادات القابلة للأمالة والذي سبب تحسن مهم في خصائص نابضيه واستقراريه مساند محركات السرع العالية. بالاعتماد على تغير لزوجة شريحة الزيت (تم أخذ ثلاث قيم) للمسند ذو الخمس وسادات، وحيث أن كل وسادة متمحورة ومثبتة لتكون قابلة للأمالة بزاوية صغيرة، وباستخدام برنامج الماتلاب، تم حساب سمك شريحة الزيت المحصورة. وبتطبيق معادلة رينولدز وحلها عددياً باستخدام طريقة الفروقات ذات الخمس عقد تم أيجاد الضغط الموزع في كل عقدة على شبكة سطح الوسادة، ليتم بعد ذلك حساب معاملات نابضية المسند. أظهرت النتائج تأثير واضح في معاملات النابضية عند تغير لزوجة المزيت. فعند نسبة لامركزية 0.3 للزوجة بين (0.04 Pas. s) و (0.058 Pas. s) كانت قيمة الزيادة في *Krr* هي 14.33 MN/m، بينما عند نسبة لامركزية 0.3 للزوجة بين (0.058 Pas. s) و (0.087 Pas. s) كانت قيمة الزيادة في *Krr* هي 11.37 MN/m. بينما عند نسبة لامركزية 0.3 للزوجة بين (0.04 Pas. s) و (0.058 Pas. s) كانت قيمة الزيادة في *Kss* هي 5.921 MN/m ، بينما عند نسبة لامركزية 0.3 للزوجة بين (0.058 Pas. s) و (0.087 Pas. s) كانت قيمة الزيادة في *Kss* هي 9.55MN/m على التوالي. في حالة نسبة لامركزية 0.3 للزوجة بين (0.04 Pas. s) و (0.058 Pas. s) كانت قيمة الزيادة في *Ksr* هي 8.95 MN/m ، بينما عند نسبة لامركزية 0.3 للزوجة بين (0.058 Pas. s) و (0.087 Pas. s) كانت قيمة الزيادة في *Kss* هي 8.19 MN/mعلى التوالي. قيم المعاملات المسيطرة الرئيسية لـ *Krr* كانت أكبر من *ksr*، كذلك فأن قيم المعاملات المسيطرة الرئيسية لـ *Kss* كانت أكبر من *Krs* لكل قيم اللزوجة التي تم دراستها. نستنتج من هذا أن التأثير السلبي لمعاملات الأزدواج المتقاطع لـ (*Ksr* ,*Krs*) قد تنتج من القيم العالية للمعاملات الرئيسية لـ (*Krr*, *Kss*) على التوالي وبهذا فأنه بالأمكان تحسين قيمة الأستقرارية لهذا المسند بتغيير لزوجة المزيت. وبهذا نوصي بأستخدام مزيتات ذات لزوجة عالية في المساند ذات الوسادات المتعددة لتحسين الأداء والأستقرارية من خلال التحكم بمعاملات النابضية للمسند.

* *Corresponding author: E-mail* : eng_ali_khalid@tu.edu.iq ; eng_ali_1977@yahoo.com