damping control. In addition, in order to generalize the present results to be useful for other QRs, the QR parameters have been nondimensionalized, and an effective nondimensional parameter through which results can be extended to other QRs is determined. The newly introduced nondimensional parameter is used against another set of data, extracted from a different QR for verification and comparative purposes. Finally, to account for uncertainties and stochastic noise in the RFD measures, each experimental stage is repeated four times; and the results of the mean behavior are reported, too.

Key Words: Quadrotor, experimental identification, frequency domain, dynamic modes, CG displacement.

R. Gavagsaz Ghoachani
r_gavagsaz@sbu.ac.ir
M. Zandi(corresponding author)
m_zandi@sbu.ac.ir
Dept. of Mechanical and Energy Engineering
Shahid Beheshti University
DOI:10.24200/J40.2019.52496.1496

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 117-127, Rview Article

- Received 4 February 2019; received in revised form 29 May 2019; accepted 9 June 2019.

Abstract

In recent years, the utilization of clean energies, especially solar energy, has increased dramatically due to their advantages over fossil fuels. This increase has led to a sharp rise in research and development activities in order to improve and optimize the energy harvesting possesses. The installation of photovoltaic power plants is the most common method for converting solar energy to electricity. However, the performance of the photovoltaic systems is extensively a function of environmental conditions such as solar radiation, ambient temperature, wind, humidity, and dust. In this regard, regions with high solar potential such as the Middle East and North Africa are favorable for the installation of photovoltaic plants. However, these regions are typically considered to be arid and deserted areas, in which the dust activities would significantly affect the performance of photovoltaic panels. In such a region, which is repeatedly faced with dust activities and aerosol dispersion, the accumulation of dust on the surface of panels is much more serious. Therefore, understanding the various dust particles removing methods from the surface of photovoltaic panels is crucial in such regions. The present study conducted a comprehensive review of the related literature in the field of photovoltaic panels cleaning methods. It was shown that these methods could be divided into four categories of natural, mechanical and electromechanical, electrostatic protection, and utilization of micro and nanoscale coatings. Furthermore, in order to provide a deeper insight into the optical and thermal modeling of photovoltaic panels and dust impacts, the related governing equations were provided. Moreover, the chemical and physical properties of accumulated dust on panels such as its size and morphology were described in detail. Investigating the related literature shows the urgent need for further research in this area, especially in the climate of Iran. The results of the current study can serve as a thorough reference for researchers, designers, and engineers who deal with photovoltaic systems in regions struggling with dust events such as the Middle East and North Africa.

Key Words: Photovoltaic, cleaning methods, dust, self-cleaning methods.

EXPERIMENTAL INVESTIGATION OF VERTICAL CG POSITION CHANGES ON QUADROTOR'S PERFORMANCE VIA FREQUENCY-DOMAIN IDENTIFICATION TECHNIQUES

M.R. Abdollahi
m.abdollahi6677@student.sharif.edu
A. Banazadeh(corresponding author)
banazadeh@sharif.edu
S.H. Pourtakdoust
pourtak@sharif.edu
Dept. of Aerospace Engineering
Sharif University of Technology
DOI:10.24200/J40.2019.52801.1503

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 129-138, Research Note

- Received 8 February 2019; received in revised form 29 May 2019; accepted 15 June 2019.

Abstract

Payload placement on quad-rotor (QR) is a subject of considerable importance during its flight and maneuvering phases. In this sense, any changes in the payload position, especially along the vertical axis, could significantly affect the QR center of gravity (CG) position and, in turn, its performance, dramatically. The current experimental study investigates the effect of payload vertical positioning on the performance and stability characteristics of a typical QR. The QR is test flown for more than eighty times with different payload positions, whose recorded flight data (RFD) are filtered using extended Kalman filter and subsequently utilized for QR frequency domain analysis. The RFDs are used to identify the QR longitudinal and lateral modes. In addition, the mode changes trend against the center of gravity location has led to the determination of the CG position at which instability occurs. The experimental results show that as the QR CG moves up along its vertical axis, its dynamics modes move towards the origin on the real axis taking the QR closer to borderline dynamic instability. Moreover, the damping behavior of the longitudinal and lateral modes with respect to CG has been extracted that in turn can lead to CG based techniques for QR

produce cold energy at different temperatures using solar energy, heat dissipation in thermal systems, biomass fuel, or other heat sources. Thermal energy is directly converted to the required mechanical energy for refrigeration in this system, which is the reason why this system is more efficient than others that convert mechanical energy to electrical energy and, then, convert the electrical energy to cold energy. In this paper, in addition to introducing and analyzing the proposed system, simulations are performed, showing that a temperature of5^0C can be produced using a solar dish at an ambient temperature of60^0C , which is suitable for saving foodstuffs and vegetables. It is assumed that the first Stirling engine produces about 60 J work from solar energy. Then, this amount of work is transferred to the second engine using a mechanical belt. The second engine produces cold energy using the transferred work. The proposed Stirling engines include two identical ST500 Gamma-type Stirling engines coupled to each other to convert heat to cool. Environmental conditions are assumed based on actual values and requirements in the south of Iran.

Key Words: Stirling engine, mathematical analysis, heat-to-cool conversion, mechatronics system, energy conversion.

THERMODYNAMIC AND ECONOMIC OPTIMIZATION OF THE REFRIGERATION SYSTEM IN A BUILDING

A. Sadeghi
a.sadeghi@gmail.com
S. Davoudabadi Farahani(corresponding author)
sdavoodabadifarahani@gmail.com
Dept. of Mechanical Engineering
Arak University of Technology
DOI:10.24200/J40.2018.51548.1474

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 107-116, Original Article

- Received 12 September 2018; received in revised form 14 November 2018; accepted 17 November 2018.

Abstract
An air conditioning system for a sample building with an area of 97.1m2 on the floor was considered. Three cooling systems are considered: a normal compression refrigeration system, a refrigeration system with an economizer, a refrigeration system with two compressors. The working fluid in three systems is R - 134 a. First, thermodynamic and economic modeling of each cycle was done, which was followed by considering the performance coefficient as thermodynamic representative and the total income requirement as economic representative. The functional pressures and the ratio of mass flow rate in the economizer are chosen as decision variables. The optimization method is Multi objective Genetic algorithm (NSGAII), and the final option is chosen using the Analytic Hierarchy Process. The economic model takes into account the cost of the components, including amortization and maintenance. In order to defb01ne a cost function, which depends on the optimization parameters of interest, component costs have to be expressed as functions of thermodynamic variables. The noteworthy point is that the cycle costs are higher than the other two cycles, and the compressor cycle costs are higher than the normal refrigeration cycle. This behavior results from the higher heating of the cycle with two compressors compared to the normal cycle. The cost of the installation of an economizer has increased the cost of the refrigeration cycle with an economizer. In this cycle, the flow rate in the evaporator is low and, to achieve the chiller load, the heat exchanger surface is higher. The cycle performance with an economizer is more than that in two other cycles, and the cost of the cycle with two compressors is higher than that of the other two cycles. In the evaluation of options, the scale of the performance coefficient is measured by 2 to 1. Based on the observed results, the normal compression refrigeration system is chosen because it is more inexpensive than other options; then, the system with an economizer is chosen as the second option.

Key Words: Multi-Objective optimization, thermodynamic and economic modeling, choice, refrigeration cycle.

A REVIEW OF DUST REMOVAL METHODS FROM THE SURFACE OF PHOTOVOLTAIC PANELS

A. Gholami
aslan.gholami@gmail.com
Sh. Eslami
shahab.eslami@gmail.com
A. Tajik
tajikaryan@yahoo.com
M. Ameri
m_ameri@sbu.ac.ir

Key Words: Numerical flow simulation, square cylinder, aerodynamic noise, splitter plate.

FREE VIBRATION ANALYSIS OF FUNCTIONALLY GRADED ANNULAR SECTOR PLATES BY THE EXTENDED KANTOROVICH METHOD

F. Fallah Rajabzadeh(corresponding author)
fallah@sharif.ir
O. Garshasbi
omidgarshasbi1990@gmail.com
A.H. Shahdadi
shahdadi_ah@mech.sharif.ir
Dept. of Mechanical Engineering
Sharif University of Technology
DOI:10.24200/J40.2018.51173.1471

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 89-98, Original Article

- Received 25 July 2018; received in revised form 21 November 2018; accepted 18 December 2018.

Abstract

In this paper, free vibration of functionally graded (FG) annular sector plates is studied using the extended Kantorovich method (EKM). To this end, based on the first-order shear deformation theory (FSDT) and Hamilton's principle, the equations of motion which are five coupled partial differential equations in terms of five displacement field variables are derived. By applying the EKM, the governing equations are reduced to two sets of ordinary differential equations in radial and circumferential directions which are solved using generalized differential quadrature method (GDQM) and state-space method, respectively. Natural frequencies are obtained through an iterative process for FG sector plates with different types of clamped and simply supported boundary conditions at the radial and circumferential edges. The results are validated by comparison with the existing ones in the literature. Finally, the effect of various parameters such as boundary conditions, material constants, and geometric parameters of the plate on the natural frequencies of FG annular sector plates is investigated. This is the first time that the EKM is used for vibration analysis of plates in polar coordinate. It is shown that the method has high accuracy and convergence for vibration analysis of plates in polar coordinate, similar to its performance in Cartesian coordinate. In addition, it is concluded that the initial guess, as the first approximation for the solution, has no effect on the final results. Furthermore, it is observed that increasing the sector angle reduces the natural frequencies while increasing the plate thickness increases them (reduces the non-dimensional natural frequencies introduced in this paper) due to the increasing plate rigidity. This effect is more pronounced in higher modes rather than the lower ones. Finally, the natural frequencies presented via a semi-analytical method for FG annular sector plates with various boundary conditions, material constants, and geometric parameters can be used as a benchmark for future studies.

Key Words: Free vibration, annular sector plates, functionally graded materials, first-order shear deformation theory, extended Kantorovich method.

PROPOSING AND ANALYSIS OF TWO COUPLED STIRLING ENGINES FOR HEAT-TO-COOL CONVERSION

M.H. Katooli
hadi.katoly@ut.ac.ir
R. Askari Moghadam(corresponding author)
r.askari@ut.ac.ir
A. Hajinezhad
hajinezhad@ut.ac.ir
Faculty of New Sciences and Technologies
University of Tehran
DOI:10.24200/J40.2019.51359.1472

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 99-105, Original Article

- Received 5 August 2018; received in revised form 15 December 2018; accepted 5 February 2019.

Abstract

The Stirling cycle is one of the thermodynamic cycles that has many advantages. Advantages of this cycle have made Stirling engines popular and widely applicable to the industry and other applications. One of the advantages of the Stirling cycle is its ability to operate inversely to produce cold energy. In this paper, a novel structure is proposed by which cold energy can be produced by a heat source using the coupling of two similar Stirling engines. By using this system, cold energy can be produced directly by the heat generated from renewable or non-renewable sources. The system is able to

NUMERICAL SIMULATION OF CONJUGATE HEAT TRANSFER IN THE RECTANGULAR COOLING CHANNEL AT SUPERCRITICAL PRESSURES

A. Ebrahimi(corresponding author)
ebrahimi_a@sharif.ir
M. Shokri
shokri_maryam@ae.sharif.ir
Dept. of Aerospace Engineering
Sharif University of Technology
DOI:10.24200/J40.2018.50479.142

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 67-77, Original Article

- Received 11 March 2018; received in revised form 17 July 2017; accepted 28 July 2018.

Abstract
In the present study, the conjugate heat transfer in a rectangular cooling channel is numerically simulated in supercritical pressure conditions. The compressible methane flow is considered as a working fluid. A finite volume scheme is utilized for the discretization of the governing equations on a collocated grid. Moreover, the central differencing scheme is employed for the discretization of the diffusion fluxes and density approximation on the control volume boundaries. Upwind and hybrid schemes are used for the density correlation approximation and the convective fluxes discretization on the control volume surfaces, respectively. An iterative solution method based on the SIMPLEC (Semi-Implicit Method for Pressure Linked Equations-Consistent) algorithm is adopted to solve the equations. The solver is developed based on the thermodynamic and transport property relations corresponding to the coolant flow conditions in the transcritical regime. The solver is validated with the experimental data of the MTP test, and the thermal behavior of methane inside the rectangular cooling channel is investigated. Moreover, a relation is derived to calculate the pseudo-critical temperature of methane according to pressure. The relative error of this relation with NIST data is less than 0.5 percent, and it operates in a range of pressure from 4.6 MPa to 30 MPa. Furthermore, the Nusselt relations presented for coolant flow with supercritical pressures are studied and corrected for the methane coolant in supercritical pressure conditions in 3D rectangular cooling channels. The relative error of modified Nusselt relations with numerical data is less than 1.0 percent.

Key Words: Conjugate heat transfer, supercritical, MTP, cooling, finite volume method, SIMPLEC, pseudo critical temperature, nusselt.

NUMERICAL STUDY OF AERODYNAMIC NOISE REDUCTION ON A SQUARE CYLINDER USING SPLITTER

S. Abbasi(corresponding author)
s_abbasi@arakut.ac.ir
M. Souri
sourimohammad.mech@gmail.com
Dept. of Mechanical Engineering
Arak University of Technology
DOI:10.24200/J40.2018.50222.1456

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 79-87, Original Article

- Received 13 March 2018; received in revised form 12 May 2018; accepted 3 June 2018.

Abstract
In this article, a numerical study of the effect of splitter plates located on the wake zone of a square cylinder on vortex formation control and aerodynamic noise reduction is presented. The flow is assumed to be unsteady and is simulated using the URANS equations and applying the turbulence model $K-\omega-SST$. Aerodynamic noise calculations are carried out by the use of lighthill analogy. For this purpose, a cylinder with a square cross-section (D = 10 mm) has been used and splitters are assumed to have a constant thickness of 2 mm with different lengths. In order to verify the numerical results, the sound pressure level obtained in different situations is compared with the experimental results, and good agreement is observed. The significant reduction of the level of sound pressure of 15% and the reduction of fluctuations with the increase of the Wake area, as well as the reduction of aerodynamic forces by applying splitter, are the results of this study. By applying a splitter with a length equal to the cylinder side, the difference in forward and backward pressure of the intermediate pressure decreases by about 40%, which results in a decrease of about 16% in the averaged drag. In this case, the Reynolds stress is reduced by about 60% as well as the pressure fluctuations by about 50%. The observation of the structure of the flow indicates that the splitter application is prevented from moving the fluid into the inner surface of the shear layers detached by the downstream splitter. The observation of the structure of the flow indicates that the splitter application is prevented from moving the fluid into the inner surface of the shear layers detached by the downstream spinner. Investigating the details of the unsteady flow structure shows that fluctuations in flow and aerodynamic forces have been reduced due to a longitudinal increase in the wake zone.

Abstract

The impact of a liquid droplet on a solid substrate and the subsequent splash-defined as the detachment of small satellite droplets from the spreading drop2014is a common everyday phenomenon. Despite splashing's ubiquity, both in our personal experience, and in such diverse applications as ink-jet printing, fuel combustion, industrial spray coating, and pesticide delivery, the mechanism of such splashes is not yet fully understood. For some applications, such as fuel combustion, splashing is beneficial, while for others, such as pesticide delivery, splashing should be minimized. For these reasons, it is important to understand the splash mechanism, and which parameters can be used to effectively control splashing behavior. The role of the compliance of soft substrates in drop splashing is important to understand it, since it is involved in many common applications, such as pesticide spray on leaves and spray cooling of flexible surfaces. When a liquid drop hits a solid surface, it often splashes and breaks into thousands of smaller droplets. Splashing occurs as two distinct types of prompt and corona. Prompt splashing takes place under the influence of surface roughness and corona splashing happens under the influence of the surrounding air of drop. In this study, splashing of droplet in impact on soft viscoelastic PDMS substrate with different film thicknesses using high-speed cameras was investigated experimentally. A mixture of ethanol / water droplets was used in the millimeter size with different physical properties for impact velocities of 3.5 to 5 m/s. Advancing and receding contact angles and roughness of substrate of different softnesses and thicknesses of the film were measured to determine the wettability of surfaces. It can be observed that, at a specified time, from the moment of impact on, the substrate with greater film thickness has lager corona height. The experimental results showed that the an increase in surface softness and thickness of the film, increases the contact angle hysteresis. Moreover, the softness and thickness of substrate have little effect on splashing / deposition threshold.

Key Words: Soft surface, droplet impact, splash, PDMS.

NUMERICAL INVESTIGATION OF THE SOLAR CHIMNEY PERFORMANCE WITH THE INTERMEDIATE ABSORBING WALL

I. Pishkar(corresponding author)
e.pishkar@pnu.ac.ir
Dept. of Mechanical Engineering
Payame Noor University

B. Ghasemi
behzadgh@yahoo.com
Dept. of Mechanical Engineering
Shahrekord University, Shahrekord

DOI:10.24200/J40.2018.10836.1454

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 57-66, Original Article

- Received 31 December 2017; received in revised form 5 May 2018; accepted 28 May 2018.

Abstract

The design of air conditioning systems is one of the effective factors in optimizing energy consumption in residential and commercial buildings. In this study, the performance of a solar chimney with different solar thermal flux on it and the use of an absorbent wall in the chimney have been investigated. Increasing the air temperature adjacent to the absorbing wall has the effect of thermosyphon in the chimney that ultimately leads to continuous air movement inside the chimney. In most of the proposed designs for these chimneys, the absorbing wall is one of the sidewalls. With sunlight warming up the wall, the air flows into the chimney. By heating the absorbing wall and increasing the temperature gradient, some heat in this wall will be lost through the conduction phenomenon in the wall thickness to the outside or inside the building. In the proposed scheme, the absorbing wall is located in the middle of the chimney and since the optimum width for the chimney is between 0.2m and 0.3m, in the proposed scheme, the distance between each wall and the intermediate absorbing wall is equal to 0.25m. In order to simulate the flow field, the equations of mass, momentum, and energy conservation are solved in the two-dimensional form with constant, incompressible, and turbulent flow assumptions simultaneously. To solve the equations, an academic code based on Fortran's language and SIMPLE algorithm is used. Due to the nature of the turbulent flow of air within the solution field, the$k-\varepsilon$ turbulent model is used because of the good performance of this model in simulating boundary layer flows with high reciprocating gradients. The intensity and concentration of heat transfer and geometric parameters related to the solar chimney, such as the entrance area, the thickness and length of the absorbing wall, and the location of the absorbing wall in the amount of discharged air flow have been investigated and the optimal values for maximum air discharge have been extracted. Moreover the absorbent wall partitioning is presented as a novel solution to increase the thermosyphon phenomenon in the solar chimney.

Key Words: Solar chimney, intermediate absorbing wall, thermosyphon, partitioning.

growth of damage and, also, healing effect, tire moving velocity has decreased. This may increase the time of loading on asphalt; therefore, rate of damage increase. In this simulations, the life of the asphalt pavement is compared with the effect of healing and without the effect. The results obtained from the simulations show that, by applying the effect of healing, the growth rate of damage and destructions caused by the permanent deformation of the pavement decreases, and the pavement lifetime can be increased up to more than 70%.

Key Words: Asphalt pavement, healing effect, damage, explicit time-discretization, finite element modeling.

NUMERICAL INVESTIGATION OF THE EFFECT OF ROUGHNESS PATTERN ON CONTACT ANGLE MEASUREMENT OF DROP ON THE SURFACE

M. Azadi Tabar
m_azaditabar@yahoo.com
F. Barzegar
ramin7barzegar@yahoo.com
M.H. Ghazanfari(corresponding author)
ghazanfari@sharif.edu
Dept. of Chemical and Petroleum Engineering
Sharif University of Technology
DOI:10.24200/J40.2018.10833.1450

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 41-48, Original Article

• Received 4 December 2017; received in revised form 1 July 2018; accepted 22 July 2018.

Abstract
Controlling surface properties, such as wetting, plays an important role in the research and development of various industries. Even though wettability has many applications in diverse technologies, such as painting, filtration, printing, porous media saturation, medicine, climate, soil, plant biology, and oil recovery enhancement, the effect of roughness pattern on contact angle and wetting needs more attention. In this study, application of the Cassie—Baxter equation for calculating the apparent contact angle of drops with different sizes on the rough surfaces is investigated. To do this, the free energy equation is analyzed on a rough and chemically homogeneous surface to study the Cassie-Baxter equation along with line tension and roughness. The contact angle calculated by using the surface fraction approximation of Cassie-Baxter for rough pattern surfaces has been compared with the values obtained from the developed numerical method. Moreover, the concept of length fraction proposed by Jaroslaw and Miller is discussed, and a numerical method is established to compute its value. To do this, spherical drops of different sizes are simulated on an artificial rough surface consisting of an array of cubic roughness. The line and surface fraction occupied by the drop are calculated and compared with the Cassie-Baxter approximation of surface fraction. Furthermore, the difference in advancing contact angle obtained by the numerical method and Cassie-Baxter model are compared. By using a developed numerical method, the length fraction and surface fraction can be computed for a wide range of drop sizes and roughness dimensions. The length fraction and surface fraction obtained in this work show oscillation behavior around the Cassie-Baxter approximation. When the drop radius tends to infinity, both values of surface and length fraction are equal to the Cassie-Baxter approximation. The results of this work help analyze/predict the apparent contact angle value for a wide range of drop sizes on rough surfaces for wettability determination.

Key Words: Wettability, contact angle, rough surface, cassie-baxter.

EXPERIMENTAL STUDY OF DROPLET SPLASHING ON VISCOELASTIC SUBSTRATE

A. Karimdoost Yasuri(corresponding author)
yasuri.am@lu.ac.ir
H. Hatami
hatami.h@lu.ac.ir
M. Shokri Rad
m.shokrirad@yahoo.com
M. Izadi
izadi.m@lu.ac.ir
Dept. of Mechanical Engineering
Lorestan University, Khoramabad
DOI:10.24200/J40.2018.10812.1443

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 49-55, Original Article

• Received 18 December 2017; received in revised form 21 April 2018; accepted 29 April 2018.

• Received 22 November 2017; received in revised form 13 August 2018; accepted 29 August 2018.

Abstract
As a kind of compact heat transfer media, open-cell foams offer a combination of advantages that include high porosity, high surface density, low pressure drop, and long-term service, making them well suited for advanced heat recovery applications. The study of fluid flow and heat transfer in porous foams is generally carried out using two models, namely the Local Thermal Equilibrium (LTE) and the Local Thermal non-Equilibrium (LTNE). The LTE model assumes no temperature difference between the local fluid and solid struts. However, when there is a significant temperature difference between solid and fluid phases, then the local phases must be treated individually by the LTNE model. The LTNE model is more difficult to apply because of more information required on the interfacial heat transfer coefficient, which is usually obtained through experimental investigations. In this study, by defining Nusselt deviation obtained by the LTE and LTNE models, operational and geometrical conditions in which the effect of LTNE model is significant are determined. As a matter of fact, based on this analysis, one can choose a suitable model for heat transfer analysis in porous open-cell foams based on the solid foam properties (such as porosity, pore density, and thermal conductivity of the foam) and the desired operating conditions (such as fluid velocity, pipe diameter, and thermal conductivity of the fluid). Results show that among the properties of porous foams, lower pore density and higher thermal conductivity (metallic foams against the ceramic foams) have the greatest impact on the importance of the LTNE model. From the viewpoint of flow operating conditions, the lower flow velocity, smaller pipe diameter, and lower thermal conductivity of the fluid increase the importance of the LTNE model in heat transfer analysis of porous foams. Among the parameters mentioned above, thermal conductivity of porous foam has the greatest impact on the local thermal non-equilibrium intensity.

Key Words: Porous foams, thermal performance, analytical solution, local thermal equilibrium, local thermal non-equilibrium intensity.

FINITE ELEMENT MODELING OF TIRE MOVEMENT ON A SMART ASPHALT PAVEMENT AND INVESTIGATION OF DAMAGE RECOVERY (HEALING) EFFECT ON ITS LIFETIME

R. Eghbalpoor
r.eghbalpoor@ut.ac.ir
M. Baghani(corresponding author)
baghani@ut.ac.ir
Dept. of Mechanical Engineering
University of Tehran
H. Shahsavari
hamid.shahal@gmail.com
M. Shojaeifard
m.shojaeifard@ut.ac.ir
S. Sohrabpour
saeed@sharif.edu
Dept. of Mechanical Engineering
Sharif University of Technology
DOI:10.24200/J.2018.10835.1452

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 33-40, Original Article

• Received 22 November 2017; received in revised form 26 August 2018; accepted 6 October 2018.

Abstract
The destruction of asphalt pavements is one of the biggest problems in road construction in the world, which incurs a huge amount of annual rebuilding. The healing process to recover damage is one of the effective methods to extend the life of these pavements, which has been presented by researchers in recent years. Self-healing polymers are a class of smart materials that, without the need for external stimulation, can repair part of the damage generated as microcracks and microvoids in their microstructures. Along with the purpose of the study, i.e., investigating the effect of healing in the lifetime of asphalt pavements, the explicit time-discrete form of the thermodynamically consistent model is presented in order to be utilized in a finite element ABAQUS software by VUMAT subroutine. The discretization method mentioned here has benefits such as low calculation costs and high reliability in response. Then, three-dimensional modeling of general vehicles' tire has been done using hyperelastic and viscoelastic materials in order to have the desired deformation on the pavement after applying the load. In the following, to evaluate permanent deformations (common failures in the asphalt pavement) and the effect of the healing process on its dilation, it is necessary to simulate its behavior in high-cycle loadings. In order to reduce the cost and time of computation, an alternative method has been used. In this method, in order to speed up the

speeds of: 50mm / min and 100mm / min were considered. The estimated values for of the unknowns by using two thermal models were different due to the assumptions are considered for models. The results obtained by two thermal models are independent of the inverse algorithms. The results show the heat flux input into the work piece as the cutting speed increases. Convective heat transfer coefficient increases with increasing the cutting speed in the first thermal model; however, this parameter decreases with increasing the cutting speed in the second thermal model. The main reason for this behavior is that, in the second thermal model, the temperature gradient in all directions (XYZ) inside the workpiece was considered. Estimated temperatures are in good agreement with measured temperatures.

Key Words: Milling process, heat flux estimation, convection heat transfer coefficient, pattern search method, nealder-mead method.

INTRODUCTION OF TWO SOLAR MSF DESALINATION SYSTEMS AND EXERGY ANALYSIS OF THEM BY “STREAM WISE” AND “SINK AND SOURCE” METHODS

H. Abdollah-Zargar
official.zargar@gmail.com
H. Ahmadi-Danesh-Ashtiani(corresponding author)
h_a_danesh@azad.ac.ir
Dept. of Mechanical Engineering
Islamic Azad University of South Tehran Branch
DOI:10.24200/J40.2018.10794.1430

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 11-21, Original Article

- Received 21 November 2017; received in revised form 3 April 2018; accepted 28 May 2018.

Abstract
According to the comparison of desalination systems and other energy consuming processes, it is observed that the energy consumption rate of desalination systems is high than that of others. Solar energy is one of the best forms of renewable energies in the world. Therefore, recently, the utilization of solar energy for setting up desalination systems has become quite widespread. Common exergy analysis method by “Stream wise” uses input and output exergy of streams assess the exergy efficiency and irreversibility. In the present study, an exergy analysis method namely “Sink and Source” is illustrated. The common exergy analysis method usually computes a system’s efficiency by more than a rational value. For example, computed exergy efficiency of a high- capacity system through the Stream wise method is 91.84%, while exergy efficiency of a high-capacity system is assessed to be 7.43% through the Sink and Source method. The inefficiency of the traditional method is also shown in the analysis of a low-capacity system. For example, the computed exergy efficiency of a low- capacity desalination system through the Stream wise method is 96.77%, while exergy efficiency of a low- capacity desalination system is 19.57% through the Sink and Source method. There is a theory about the reverse relation of utility fluid temperature and exergy efficiency of systems. The validation of the mentioned theory is measured by Sink and Source exergy analysis method. In the following, in order to reduce energy consumption of the analyzed system, a solar field is suggested for a high-capacity system. Due impossibility of implementing of the mentioned solar field in Iran and the low efficiency of high-capacity desalination plants, a new low-capacity desalination plant is suggested by the imposition of little changes on operating parameters of the high-capacity system. The presented low- capacity desalination system has higher exergetic efficiency and is capable of working with non-concentrating solar collectors. Finally, the proposed argument is validated based on the obtained results, and it is shown that the Stream wise method loses its functionality in some conditions.

Key Words: Exergy analysis, solar assisted, MSF desalination system, stream wise, sink and source.

ANALYTICAL STUDY OF THE EFFECT OF LOCAL THERMAL NON-EQUILIBRIUM INTENSITY IN POROUS FOAMS

A. Abdollahipour
a.abdollahipour@mail.sbu.ac.ir
J. Aminian(corresponding author)
j_aminian@sbu.ac.ir
Dept. of Mechanical and Energy Engineering
Shahid Beheshti University
DOI:10.24200/J40.2018.10832.1451

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 23-32, Original Article

SIMULTANEOUS ESTIMATION OF HEAT FLUX INTO WORK PIECE AND CONVECTIVE HEAT TRANSFER COEFFICIENT IN MILLING PROCESS

S. Davoodabadi Farahani
sdavoodabadifarahani@gmail.com
Dept. of Mechanical Engineering
Arak University of Technology
DOI:10/24200/J40.2018.10802.1433

Sharif Mechanical Engineering Journal
Volume 35, Issue 2, Page 3-10, Original Article

- Received 6 August 2017; received in revised form 13 May 2018; accepted 20 May 2018.

Abstract
This study investigates the simultaneous estimation of heat flux input into a work piece and convective heat transfer coefficient in milling operations. The material of the work piece is AISIH13. Temperatures in 5 points inside the work piece were measured using thermocouples(K-type). Two thermal models for the work piece were considered to solve the direct problem. Work piece material is an isotropic and the thermal property of work piece is constant. In the first thermal model, it is assumed that temperature changes only with time. This model uses the mean temperature of all thermocouples. For direct problem solving, the code of this model is written in MATLAB software. The second thermal model is a transient 3D problem and temperature inside work piece changes with time and location. This model uses the temperature of each thermocouple. For direct problem solving, the code of this model is written in ANSYS software. Heat flux and convective heat transfer in two thermal models are unknown. Thus, the inverse heat transfer method is used to estimate the unknowns. This problem will not be solved by standard inverse algorithms. Thus, pattern search algorithm and Nealder-Mead method are used. For both of algorithms, MATLAB toolbox was used. In this study, two cutting