were subjected to projectiles with different impact velocities of 400, 450, 509 and 550 m/s. Results of the FE-Analysis runs conducted in this paper indicate that the response mitigation of a composite armor is significantly affected by both the density and thickness of its PU-foam layer. For a given thickness of foam layer, it was observed from the FE-analysis runs that the optimum value for the foam-density that results in the maximum response mitigation is approximately 150 kg/m3. The impact pressure that is dissipated by a composite armor is significantly decreased as the foam-density diverts from its optimal value. For a given foam-density, it was found that the response mitigation of the armor was improved with an increase in the thickness of the foam layer. For the composite armors investigated in this study, the rate of improvement in the response mitigation with increased foam-layer thickness was found to be very slow for the thickness values beyond 5 mm. The impact velocity of the projectile, as an indicator of strain rate, was also investigated in this paper. The dynamic response of foam layers of relatively lower thickness and density was found to be more sensitive to the variations of strain rate.

Key Words: Polyurethane foams, density, thickness, projectile impact, layered armor, strain rate.

NON-CLASSICAL THERMOMECHANICAL ANALYSIS OF ND:YVO4 SLAB LASER UNDER MULTI-PULSE END PUMPING USING FINITE ELEMENT METHOD

H. Zarei
hamidreza90.zarei@gmail.com
M. Mojahedi(corresponding author)
mojahedi62@gmail.com
Dept. of Mechanical Engineering
Golpayegan University of Technology
DOI:10.24200/J40.2018.6411

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 125-132, Original Article

- Received 19 June 2017; received in revised form 23 August 2017; accepted 5 September 2017.

Abstract

In order to create a laser beam, one common method is to use beam pumping. In this method, beam is firstly emitted to the active medium of laser crystal and the electrons in the last shell of atoms are stimulated. With some internal interactions, laser beam will be created and some pumping energy is converted to heat in the crystal. In solid-state lasers, large amounts of pumping power may lead to heat generation in the active laser medium and a non-uniform temperature distribution in the crystal is generated. The temperature gradient causes thermal stress in the crystal structure and can be effective on thermomechanical behavior of the crystal and the quality of the laser beam. With increasing the pumping power, the heat generation within the active media is increased and consequently the thermal effects including the thermal lensing, polarization of output beam, mechanical stress, reducing the efficiency of output beam, are intensified. Even in this case, thermal stress causes failure. Hence to achieve a high quality beam laser, it is necessary to investigate the thermomechanical behavior of the laser crystal, through the designing of solid state lasers. In the past works, thermomechanical behavior of the crystal was investigated in framework of Fourier heat conduction theory (Heat wave diffusion speed is considered to be infinite), while this theory is not appropriate to study the heat conduction for medium under pulsed heat with appreciable relaxation time. Hence, in this research, by providing a finite element formulation based on the non-Fourier thermoelasticity (with considering relaxation time), distributions of temperature and stress in the Nd:YVO4 crystal are calculated. The results show that the maximum temperature and stress calculated by non-Fourier theory, are larger than those calculated by the Fourier theory and also the non-Fourier theory predicts lower failure power for crystal. In addition, if the relaxation time is lower than a certain value, the results of classical and nonclassical theories are closely equal to each other.

Key Words: Pulsed end-pumping, non-fourier, finite element, Nd:YVO4.

THE PERFORMANCE OF RIGID POLYURETHANE FOAMS IN DAMAGE MITIGATION OF STRUCTURES DUE TO THE IMPACT OF A SUPERSONIC PROJECTILE

M.H. Andami
hosein.andami@yahoo.com
H. Toopchinezhad(corresponding author)
h.toopchinezhad@razi.ac.ir
Faculty of Mechanical Engineering
Razi University, Kermanshah
DOI:10.24200/J40.2018.6420

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 133-140, Research Note

- Received 3 January 2017; received in revised form 3 June 2017; accepted 12 June 2017.

Abstract

In this paper, the performance of the rigid polyurethane (PU) foams used in composite armors to mitigate the impact pressure and energy of a projectile has been studied numerically through finite element methods. The armors of this study comprise two layers, namely, a metal layer and a PU-foam layer. A sophisticated FE-Model was developed to simulate the dynamic performance of the composite armor under the impact of a projectile using the software package ANSYS-Autodyn. The FE-Model was verified with the results of a previous experimental study in which a projectile impacted a certain two-layer armor comprising a steel front face that was attached on a layer of PU-foam. The main objectives of this research study are to study the effects of design parameters such as the thickness and density of the PU-foam and the impact velocity of the projectile (representing the strain rate) on the performance of a composite armor. To achieve the research objectives, PU-foams of different densities of 80, 160, 288 and 320 kg/m3, at six different thicknesses of 1.25, 2.5, 5, 7.5, 10 and 12.5 mm were modeled. Additionally, the armors

• Received 30 April 2017; received in revised form 24 July 2017; accepted 8 August 2017.

Abstract

Due to important rule of edge stresses in the delamination and local failure of laminated structures, the objective of this work is to analyze the interlaminar stresses in laminated hybrid composite cylindrical shell which is subjected to centrifugal body force using layerwise theory of Reddy (LWT). The model is under centrifugal forces caused by rotation of the cylinder about its axis. The principle of minimum total potential energy is used to obtain the governing equations of the rotating cylinder. The layerwise theory (LWT) of Reddy is employed for discretization of the governing equations to a system of ordinary differential equations in terms of the displacement of numerical surfaces. To solve the equations of motion of the cylinder in the LWT, the equation is written in the decoupled form by definition of modal space variables. An analytical solution is presented for governing equation. The solution is completed by imposing the boundary conditions in the edges of the cylinder. The interlaminar and in-plane stresses are obtained using the displacement field of layerwise theory for the free and clamped boundary conditions. The interlaminar stresses in the Hybrid cylindrical shell are obtained using two different methods, the constitutive law of the numerical layers, and integrating the equilibrium equations of motion. To validate the results, the results of LWT for cross-ply cylindrical shell are compared to those of Finite element solution in the commercial finite element code Ansys. The results showed an excellent accuracy of the layerwise theory in calculating the interlaminar and in-plane stresses in hybrid cylinder. Various figures from the distribution of the stresses along the interfaces of layers and through the thickness of rotating cylinder are presented in the numerical results. The effects of the geometry, the types of layering, the speed of rotation and hybridation of laminated cylindrical shell on the interlaminar stresses are analyzed. Consequently, the delamination of the layered can be predicted and prevented.

Key Words: Laminated hybrid composite, out of plane stresses, cylindrical shell, layerwise theory, cross-ply layer stacking.

THE EFFECTIVE PARAMETERS ON AUTOMATIC LINE-BREAK CONTROL VALVE IN GAS TRANSPORTATION PIPELINES

M. Mahmoodi
mehdymahmoody@gmail.com
M. Gorji Bandpy(corresponding author)
girji@nit.ac.ir
Faculty of Mechanical Engineering
Babol University of Science and Technology
DOI:10.24200/J40.2018.6412

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 115-123, Original Article

• Received 7 May 2017; received in revised form 14 August 2017; accepted 5 Septemebr 2017.

Abstract

Automatic line-break control valve can be used in zones with no access to national electricity network, hazardous or impassable area enclosure, for passive defence conditions or to protect different ecosystems such as rivers and ground water, forests and fertile land and etc. The pipeline break causes a growth in oil velocity magnitude and fluid pressure reduction consequently but the pressure drop is not a good signal for line break detection because it is low, therefore reference tank is used. The reference tank is connected to the pipeline through a calibrated orifice with check valve. The tank pressure is higher than pipeline pressure when pressure reduction occurs by line breaking. If the pressure difference between tank and pipeline is higher than the sustainable pressure value of diaphragm valve, the valve will be closed by actuator. In this paper, the effects of orifice diameter, pipeline initial pressure and pipeline break pressure drop rate on the differential pressure of diaphragm valve in automatic line-break control valve have been studied with nitrogen by 27 experiments. The differential pressure of diaphragm valve is increased by the growth of pipeline break pressure drop rate. The occurrence time of maximum differential pressure of diaphragm valve just depends on orifice diameter. The maximum of differential pressure increases with decrease of orifice diameter. The differential pressure of diaphragm valve increases by decrease of pipeline initial pressure for constant orifice diameters and pipeline break pressure drop rates. The curves of maximum differential pressure generated by different pipeline break pressure drop rates are shown for different orifice diameters and pipeline initial pressure. The values of curves in this diagram with 10 percent safety factor can be used in automatic line-break control valve setting which are installed on gas transportation pipelines because of the nitrogen gas using instead of the natural gas.

Key Words: Line-break, control valve, orifice diameter, pressure drop rate, experimental study.

of economic, thermal storage tank with minimum investment cost rate (13.24 $/h) has the maximum exergy destruction cost rate within 1724.13 $/h and 1783.37 $/h respectively for summer and winter while the collector. Advanced exergoeconomic analysis indicates that the substantial portion of its avoidable part is exogenous indicating that the technological improvement of remaining components lead to the decrement of the exergy destruction cost rates of the auxiliary boiler. Moreover, the maximum value of the investment cost rate is related to the collector containing the maximum endogenous avoidable part with values of 646.465 $/h for summer and 455.015 $/h for winter. Similarly, in exergoenvironmental analysis, auxiliary heater with the lowest investment environmental impact has the maximum environmental impact associated with the exergy destruction with values of 242.11 Pts/h and 306.95 Pts/h respectively for summer and winter due to the utilization of the natural gas. Advanced exergoenvironmental analysis, the auxiliary heater has the maximum exogenous avoidable environmental impact associated with exergy destruction indicating the improvement of other components has a positive effect on decreasing of its environmental impact.

Key Words: Kalina cycle, advanced analyses, exergoeconomic, exergoenvironment, solar energy.

SIMULATION ANALYSIS OF POLYMER NANOCOMPOSITES REINFORCED WITH CARBON NANOTUBES AND COILED CARBON NANOTUBES

E. Yousefi
ensieh.yousefi@student.kuleuven.be
M. Baniassadi(corresponding author)
m.baniassadi@ut.ac.ir
M. Mahdavi
mostafa.mahdavi@gatech.edu
M. Baghani
mostafa.mahdavi@gatech.edu
Gh. Faragi
ghfaraji@ut.ac.ir
Dept. of Mechanical Engineering
University of Tehran
DOI:10.24200/J40.2018.6418

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 95-102, Original Article

- Received 13 March 2017; received in revised form 3 June 2017; accepted 1 July 2017.

Abstract

Carbon nano tube (CNT) and coiled carbon nano tube (CCNT) have exceptional properties and can improve mechanical, electrical and thermal properties of nano composites. Among carbon nanostructure, a great deal of interests has been recently focused on CCNTs due to their specific properties which stem from the specific helical structure, which this field is one of the most interesting fields in engineering. In this paper, a multi-scale model is developed to study the mechanical properties in polyethylene reinforced by CNTs and CCNTs. Influence of volume fraction (VF) and geometric of the nano particle that is in two types of the CNT and CCNT, on overall stress - strain response have been investigated. Elastic modulus for each sample is calculated based on the linear part of stress-strain curve, and compared to each other. Results show with adding 1 percent of the CNT and CCNT to the polymer elastic modulus increases 17.5% and 9.5%, respectively, by using 1.5 percent of the CNT elastic modulus increase 21.4%. As a result, when the volume fraction increases the strength of nano composites enhances. In this paper also effect of orientation of CNT in matrix has been studied. Samples with 1 percent volume fraction in two isotropic and anisotropic states are generated and mechanical properties of these samples are studied. Results show a huge reduction in the stress-strain response of the anisotropic composite compared to the isotropic composite. Also effect of coil shape of CCNT were studied and compared with straight CNT. According to the results straight CNT is a better reinforcement compared to CCNT and as the coil angle increases, the overall stress-strain response improves as well.

Key Words: Nanocomposite, carbon nanotube, coiled carbon nanotube, mechanical properties.

BOUNDARY LAYER ANALYSIS IN ROTATING CROSS-PLY HYBRID COMPOSITE CYLINDER WITH DIFFERENT EDGE CONDITIONS

I. Ahmadi(corresponding author)
i_ahmadi@znu.ac.ir
M. Najafi
mahsanajafi@znu.ac.ir
N. Ataee
neda.ataee@znu.ac.ir
Dept. of Mechanical Engineering
University of Zanjan
DOI:10.24200/J40.2018.6408

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 103-114, Original Article

ELASTIC-PLASTIC PIN-ENDED BEAMS UNDER IMPULSIVE LOADING WITH REGARDING THE LINEAR HARDENING EFFECTS

M. Shams Alizadeh(corresponding author)
mshalizadeh@gmail.com
K. Heidari Shirazi
k.shirazi@scu.ac.ir
Sh. Moradi
moradis@scu.ac.ir
H.M. Sedighi
h.msedighi@scu.ac.ir
Dept. of Mechanical Engineering
Shahid Chamran University, Ahwaz
DOI:10.24200/J40.2018.6416

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 71-80, Original Article

- Received 14 December 2016; received in revised form 3 April 2017; accepted 9 May 2017.

Abstract

The Counter-intuitive behavior which means that the permanent deflection of the elastic-plastic beam come to rest in opposite direction of the impulsive loading, normally appears and disappears abruptly, in certain small ranges of the loading and structural parameters. One of the most important issues in the study of this phenomenon is the determination of the influence of the different parameters on this behavior. This present paper studied the effects of hardening in the counter-intuitive dynamic behavior of the elastic-plastic pin-ended beams under impulsive loading, by developing the proposed Galerkin's numerical model and by presenting a novel algorithm. Galerkin's method as well as Finite element code ANSYS/LS-DYNA were applied to study this phenomenon and the results of these two methods have been compared. In order to regard the hardening effects in Galerkin's method, a new algorithm was proposed. To test the validity of our proposed algorithm, the response of the elastic-perfectly plastic beam was firstly studied, and the hardening effects on the counter-intuitive response was investigated afterwards. The displacement-time history curves of mid-span of the beam were studied in detail and the region of the occurrence of the counter-intuitive behavior was determined. Furthermore, using the finite element code, energy diagrams of the beam were also investigated. It was been found out that the counter-intuitive response is a phenomenon which is very sensitive to loading so that it may appear with a little change in the amount of loading. The results also showed that when considering the hardening effects, both methods predicted two continuous and distinct regions of loading for the occurrence of this phenomenon where the first had a narrower band and occurred in the tight region of the impulsive loading while the next one had a wider band and occurred in the vast region of the impulsive loading. In addition, our investigations on the energy diagrams showed that, this anomalous behavior would occur in the proper proportions of kinetic to internal energy.

Key Words: Counter-intuitive behavior; anomalous response; impulsive loading; dynamic elastic-plastic analysis.

MODELING AND ADVANCED EXERGY, EXERGOECONOMIC AND EXERGOENVIRONMENT ANALYSES OF NEW CCHP KALINA CYCLE INTEGRATED WITH SOLAR PARABOLIC TROUGH COLLECTORS

M. Rahmatian
m_rahmatian69@yahoo.com
F. Ahmadi Boyaghchi(corresponding author)
fahmadi@alzahra.ac.ir
M. Sabaghian
mah.sabaghian@gmail.com
Dept. of Mechanical Engineering
Alzahra University
DOI:10.24200/J40.2018.6407

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 81-93, Original Article

- Received 8 March 2017; received in revised form 24 June 2017; accepted 21 August 2017.

Abstract

In this research, a Kalina system integrated with the parabolic trough solar collector is proposed to produce power-cooling load and power-heating load for summer and winter, respectively. The proposed system is modeled using the conventional and advanced exergy, exergoeconomic and exergoenvironmental analyses. Based on exergy modeling, in summer collector and in winter auxiliary heater are the worst components respectively with 19629 kW and 11561 kW exergy destruction rate while according the advanced exergy analysis the major portion of their exergy destructions are related to the endogenous part while the avoidable endogenous parts are zero. On the other hand, the avoidable exogenous subdivision of the collector in summer mood and auxiliary boiler for winter are significant with values of 4492.84 kW and 2676.31 kW, respectively. From the viewpoint

geometric nonlinearity significantly affect the mechanical behavior of the system. Therefore, neglecting the abovementioned effects leads to errors in the instability prediction of microswitches. Most of microswitches consist of a microcantilever with a proof mass and a fixed substrate which there is an air gap between them. By applying voltage to the system, the microcantilever starts to deflect into the fixed substrate. In this paper, pull-in instability and deflection of MEMS switches are investigated based on the size dependent model. The nonlinear model is introduced by considering modified couple stress theory and fringing field effects as well as geometric nonlinearity. Utilizing the minimum total potential energy principle, the static equation of motion is derived in framework of the nonclassical theory. The effects of various parameters on static pull-in instability are studied and errors of considering the linear model or classical theories is calculated. The results show that the presented model is capable to predict the displacement and pull-in instability of the microswitches.

Key Words: MEMS, static pull-in instability, modified couple stress theory, fringing field, large deflection.

NUMERICAL INVESTIGATION OF THE EFFECT OF TWO-BLADEDROTARY TUBE-INSERT ON PERFORMANCE INSTANT GASLIGHT WATER HEATERA

F. Eghlimi-Moghadam
fatemeh.eghlimi@gmail.com
K. Goudarzi(corresponding author)
kgoudarzi@yu.ac.ir
Gh. Zendeh-Budy
zendehbudi@yahoo.com
A. Moosaie
moosaie@yu.ac.ir
Dept. of Mechanical Engineering
Yasouj University
DOI:10.24200/J40.2018.6417

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 63-69, Original Article

- Received 21 November 2016; received in revised form 7 January 2017; accepted 19 June 2017.

Abstract

In this study, the effect of tube-insert as a passive method to increase heat transfer, instant gaslight water heater performance was evaluated. The simulation results revealed that for rotary tube-insert with different cross section shapes. The article present a numerical study of turbulent heat transfer and flow characteristics inelliptical tube equipped with two-bladed rotary tube-insert. Seven different tube-insert are tested in the current work; they included the two-bladed tube-insert with four different length of pitch 75, 100, 150 and 200 mm and three blade angle 40, 55 and 70 degree. In this study, the effect of tube-insert as a passive method to increase heat transfer, instant gaslight water heater performance was evaluated. To analyze the efficiency and effectiveness tube-insert geometric parameters and physical parameters of the flow, steady and incompressible flow gaslight water heater, using the commercial software ANSYS CFX 14 was modeled. Turbulent flow regime of the fluid is used. The numerical simulation is performed using SST turbulence model and wall function near the walls in ANSYS CFX to study the phenomena of flow field in a elliptical tube with helical blade rotors. The SIMPLE algorithm pressure-velocity coupling method, the standard pressure, the high resolution scheme discretization scheme for momentum, energy are selected in the simulation. The results of the simple tube compared with the theoretical Gnielinski and Petukhov relationship for validate of results. Using modeling results, performance parameters such as the coefficient of friction, heat transfer coefficient, and thermal performance has been calculated and analyzed. Effect of Tube-insert geometric parameters such as the length and angle of blades were evaluated on performance gaslight boilers. Reynolds numbers range from 3580 to 14320 is investigated. The average coefficient of thermal performance for gaslight water heater equipped with two-bladed tube-insert with length of pitch 75, 100, 150 and 200 mm, respectively, 1/017, 1/027, 1/033 and 1/042have been. By observing the results and compare them two-blade tube-insert with blade angle 55 and 70 degrees with an average coefficient of thermal performance 1/215 and 1/193 best performance in gaslight water heater have been studied. Therefore, the use of rotary tube-insert as a passive method can significantly increase heat transfer in the heat exchangers. The increase in flow velocity of the heat exchanger tube equipped with a rotary tube-insert reduce friction and prevent the accumulation of fouling. This method can also be used in heat exchangers.

Key Words: Rotary tube-insert, gaslight water heater, heat transfer, friction factor, nusslet number, thermal performance factor.

NUMERICAL ANALYSIS OF THE COUNTER-INTUITIVE DYNAMIC BEHAVIOR OF THE

NUMERICAL STUDY OF STREAMING IN THERMAL BUFFER TUBE

M. Bahrami(corresponding author)
bahramymohsen@modares.ac.ir
M. Karimi
karimi.mhs@gmail.com
K. Ghorbanian
ghorbanian@sharif.edu
Dept. of Aerospace Engineering
Sharif University of Technology
DOI:10.24200/J40.2018.6413

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 41-51, Original Article

- Received 5 September 2016; received in revised form 15 May 2017; accepted 27 May 2017.

Abstract

Streaming appears in almost all traveling wave thermoacoustic engines and refrigerators. This phenomenon may increase or decrease the efficiency of thermoacoustic devices. It should be emphasized that properly designed streaming may also be used to eliminate the required heat exchanger in the abovementioned devices. However, in most cases, due to the lack of the relevant knowledge, this phenomenon is considered as a non-preferred occurrence and a source of secondary losses. Thermal buffer tube separates the hot and ambient heat exchangers in travelling-wave thermoacoustic engines as well as the cold and ambient heat exchanger in pulse-tube refrigerators. The role of the thermal buffer tube is to pass the acoustic energy while minimizing the heat transport due to the boundary-layer entropy flow, heat conduction through the gas and the tube walls, radiation, and mass streaming. The latter is caused by finite-amplitude acoustic oscillations and can transport significant amount of heat between heat exchangers. The heat transport depends on temperatures at the ends of the thermal buffer tube. Acoustic motion of gas parcels between nearly isothermal environment in the heat exchanger and nearly adiabatic space in the tube leads to effective temperature jumps at the tube boundaries.

The main goal of this paper is to numerically investigate the periodic field in thermal buffer tubes with axisymmetric geometry of thermoacoustic Stirling heat engines. The effects of variable geometry (tapered and uniform cross-section tube) and turbulence on the streaming intensity in viscous compressible oscillating flow is studied. First, one-dimensional wave equation based on Rott's assumptions in thermal buffer tube is solved by using DeltaEC software. The, hereby determined boundary and initial conditions are employed to investigate the processes by a commercial numerical solver. The simulation is carried out for straight and tapered axisymmetric tube with a pressure-based PISO solver. The results indicate that one may control the streaming effect by changing the duct geometry. Besides, it's showed that turbulence and gravity have an effective roll in reducing secondary flow.

Key Words: Streaming, thermoacoustic Stirling engine, thermal buffer tube, numerical simulation.

STATIC INSTABILITY ANALYSIS OF A NONLINEAR MICROSWITCH CONSIDERING MODIFIED COUPLE STRESS THEORY

M. Mojahedi(corresponding author)
mojahedi62@gmail.com
S. Hakamiha
sepehrhakamiha@gmail.com
Dept. of Mechanical Engineering
Golpayegan University of Technology
DOI:10.24200/J40.2018.6410

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 53-61, Original Article

- Received 19 October 2016; received in revised form 3 April 2017; accepted 9 May 2017.

Abstract

Microelectromechanical systems (MEMS) are used in many fields of industry like automotive, aerospace and medical instruments. Among the various ways to operate the MEMS devices, the electrostatic actuator is the common mechanism, due to simplicity and fast response. Previous experiments have shown that the mechanical behavior of devices, which their sizes are in order of micron and submicron, are dependent to size dependency. They also have illustrated that by decreasing the dimension of structures, the size dependent effect is highlighted. In this case, the classical theories are not capable to predict the size dependent effects and mechanical behavior of the microstructures properly. Therefore, nonclassical theories such as modified couple stress and strain gradient theories have been introduced. It was shown that the modified couple stress theory can accurately predict the size dependent behavior of microstructures. There are some influences observed in the MEMS, that they have notable effects on the mechanical behavior of microswitches, such as fringing fields and large deflection. When the air gap is larger than the electrode's width of microswitches, the impacts of fringing fields and

is very high and should be considered. Using numerical simulations, a method to find a suitable design considering both the acoustic and aerodynamic performance presented. In this method, at First air supply channel system of marine gas turbine according to the requirements of the aerodynamic and acoustic were categorized into three sections; input, plenum chamber and output channels with circular cross sections. The geometric dimensions of input and output channels determined using the theory of plane waves within the channel, without considering the effects of flow also considering the limits of space inside the float at dominant frequency. Then suitable size of the intermediate cross section of the channel in terms of both aerodynamic and acoustic requirements with regard to the effects of flow using numerical methods were studied and determined. Various 3 Dimensional turbulent flows inside chamber are considered for this work where large eddy simulation turbulence model is used. Ffowcs, Williams and Hawkings model is used for the sound propagation process based on Lighthill integral equation. Validity of the simulation is checked by comparing results (sound pressure level) against experimental data in a chamber and error is acceptable over range of studied frequencies. The results showed that channel system aerodynamic performance decreases and its acoustic performance initially increases and then decreases with the increase in cross section of the plenum chamber than the cross section of the inlet / outlet channel. In addition, deviation from plane waves in considering the effects of flow is observed in channel. That is due to the effects of the current flowing through Channel system and the effect of quadrupole sources in the production of sound in the channel system, which causes higher modes.

Key Words: Aero acoustics , large eddy simulation, lighthill analysis, ffowcs williams and hawkings models , propagation of wave in duct.

STATISTICAL CORRELATION FUNCTIONS AND RECONSTRUCTION OF MICRO/ NANOCOMPOSITES; A REVIEW

M. Mahdavi
mostafa.mahdavi70@gmail.com
E. Yousefi
nc.yosefi@ut.ac.ir
M. Baniassadi(corresponding author)
m.baniassadi@ut.ac.ir
Dept. of Mechanical Engineering
University of Tehran

DOI:10.24200/J40.2018.6419

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 31-40, Original Article

- Received 31 August 2016; received in revised form 22 February 2017; accepted 12 June 2017.

Abstract

N-point correlation functions are used for calculating properties of heterogeneous systems. Power and main advantage of statistical continuum technique is the direct link to statistical information of microstructures. Two-point correlation functions are the lowest order of correlation functions that can describe the relationship between morphology and properties of microstructures. Statistical pair correlation functions are achieved by scattering electron microscopy or small-angle X-ray scattering experimentally. Higher order correlation functions must be calculated or measured to increase the accuracy statistical continuum approach. Microstructure two-point correlation functions are well-known class of statistical descriptors for characterization and reconstruction of heterogeneous microstructures. In this study, a comprehensive review on statistical correlation functions that are used in micro and nanostructures is done and then, focused on reconstruction of these structures by using numerical and experimental approaches. Description and characterization of heterogeneous systems had high importance for scientists for last decades and different approaches have been developed for determining 3D descriptors of heterogeneous systems. Statistical continuum mechanics provided alternative approach for reconstruction and characterization of heterogeneous systems. Reconstruction approaches have been improved by developing of numerous simulation techniques in recent years. Anisotropic structures, orientation distribution, shape, and geometrical features can be extracted from statistical correlation functions. 3D reconstruction of microstructures and nanostructures is a new way to investigate the behavior of these structures precisely. In this method the distribution of nanoparticles is determined, and their coordinates are saved which is very helpful for determination of their treatment. Mani different researches have used reconstruction approach to enter their experimental samples to finite element software. Then, by applying the properties of each phase they investigate about different behavior of their experimental samples. Depended on samples different approaches are used for reconstruction and researches have tried different ways to reduce reconstruction error. In this study, different approaches are discussed, and the calculation of error is presented.

Key Words: Statistical correlation functions, polymer nanocomposites, reconstruction.

software by using constant-rate mass transfer model to study the effects of different steam and water flow parameters, including water subcooling degree, Reynolds number of water flow, and steam mass flux, on dimensionless steam plume length. Moreover, constant-rate mass transfer model was revised by using the results obtained from the simulation and a new correlation for calculation of the constant coefficient of the mass transfer model was proposed as a function of flow parameters that influence the phenomenon to increase the accuracy of future simulations. The proposed correlation agrees well with numerical data and most of the data lie in the range of ±20% of the correlation.

Key Words: Two-phase flow, CFD, direct contact condensation, steam jet.

MODIFICATION OF A PLASMA ACTUATOR MODEL FOR INDUCED WALL-JET SIMULATION

A. Ebrahimi(corresponding author)
ebrahimi_a@sharif.edu
J. Omidi
jomidi@alum.sharif.ir
Dept. of Aerospace Engineering
Sharif University of Technology
DOI:10.24200/J40.2018.6409

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 11-17, Original Article

- Received 26 July 2016; received in revised form 8 March 2017; accepted 9 April 2017.

Abstract

Recently, active flow control by dielectric barrier discharge (DBD) plasma actuators has been increased. These actuators are known as more efficient, low-cost, without the need of moving parts, low power consumption, small size, low weight, easy installation and without delay in control. All these features have attracted researchers to use this actuator in a variety of cases, such as turbulence flow control, laminar to turbulent transition suppression, separation control, drag reduction and mitigation of noise pollution. However, most of the studies suffer from lack of an accurate numerical model which can simulate this phenomenon in details. Computational analysis of this phenomenon is very complex and difficult due to a combination of ionization phenomena and the interaction of the fluid flow with the actuator effects. For the exact solution of this phenomenon in certain conditions, Maxwell and Navier-Stokes equations must be combined, while this non-linear solution combination will be very difficult. One of the models for modeling the interaction between the actuator and fluid flow is an electrostatic model which adds the actuator effect as source terms in the momentum equations by solving the electrical potential equation and charge density equation. In this study, an improvement is proposed to enhance the simulation accuracy of a model used for plasma actuator effect under interaction with fluid flow. In the modified model suggested, a boundary condition for charge distribution on the charged surface is presented based on a relationship between the independent electrical potential and charge density equations. Further, semi-empirical relations are utilized to calculate the produced plasma extend. The effect of the actuator on induced jet shows a good agreement with experimental results and does not need experimental tests for parametric calibration.

Key Words: Electrostatic model modification, DBD plasma actuator, fluid flow control.

DESIGN AND NUMERICAL SIMULATING OF THE PERFORMANCE OF ACOUSTIC PLENUM CHAMBER OF THE MARINE GAS TURBINE AIR SUPPLY SYSTEM

M. Rasoulimoghadam
mehrdad.rasoulimoghadam@yahoo.com
M. Agha Seyyed Mirzabozorg
mirzabozorg@mut-es.ac.ir
S. Kheradmand(corresponding author)
saeid_kheradmand@yahoo.com
Dept. of Mechanical and Aerospace Engineering
Malek-Ashtar University of Technology
DOI:10.24200/J40.2018.6414

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 19-29, Original Article

- Received 27 July 2016; received in revised form 15 July 2017; accepted 5 September 2017

Abstract

In the present work, cross section size investigation of the plenum chamber of a marine gas turbine air supply system was done. Flow in ducts make noise and in the case of turbine inlet due to amount of flow this noise

Abstracts of Papers in English

MODIFYING "CONSTANT-RATE" MASS TRANSFER MODEL FOR CFD ANALYSIS OF DIRECT CONTACT CONDENSATION OF STEAM JET IN WATER FLOW

S. Vedadi Kalantar
saeedkalantari89@gmail.com
Dept. of Energy Engineering
Sharif University of Technology
A. Esmaeeli
ali_esmaeeli_67@yahoo.com
S. Zare
saman.zare@stu.um.ac.ir
M. Pasasndideh Fard(corresponding author)
mpfard@um.ac.ir
Dept. of Mechanical Engineering
Ferdowsi University of Mashhad
DOI:10.24200/J40.2018.6415

Sharif Mechanical Engineering Journal
Volume 34, Issue 2, Page 3-10, Original Article

- Received 5 March 2016; received in revised form 3 April 2017; accepted 18 April 2017.

Abstract

Direct contact condensation is used in industrial applications, such as steam jet pumps, direct-contact heat exchangers and nuclear reactor cooling systems, due to its highly efficient heat and mass transfer. When steam is injected into subcooled water, a steam plume is generated at the exit of the steam nozzle, surrounded by an interface around the steam plume. The direct and quick transfer of heat, mass and momentum across steam-water interface makes the physics of direct contact condensation very complex. To efficiently design the above mentioned equipment of direct contact condensation, a proper understanding of heat and mass transfer in this phenomenon is required. Several experimental and theoretical works have been performed on steam jet condensation in water. However, there have been a few numerical investigations of this phenomenon. Different heat and mass transfer models have been used by researchers to develop a suitable numerical tool to simulate steam condensation process. Constant-rate mass transfer model is one of the models used in numerical simulation of direct contact condensation in steam injection into water. This model needs a specific empirical coefficient for each simulation. In this study, considering available experimental results of previous investigations, a numerical simulation was performed in ANSYS Fluent