Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 129-135, Research Note

- Received 19 May 2014; received in revised form 17 November 2014; accepted 14 December 2014.

Abstract

Pull-in is a critical case which occurs in many micro electromechanical devices (MEMS) such as sensors, actuators and micro-electronic switches. Determination of the voltage at which Pull-in has occurred can determine the capacity of such devices. In literature, there is some work which has obtained the value of this voltage using lamped simulation or experimental analysis. In this paper, at first, the equation of voltage-displacement for transverse deflection of a circular clamped micro-plate, with a single or dual back plate, based on Von-Karman and classical plate theory, has been obtained. Then, analytical relations for static Pull-in voltage that has caused instability and damage to the system have been derived. These relations show that the value of Pull-in voltage obtained by these relations is about 1.3 times that calculated by previous researchers who used distributed mass simulation for validation of the obtained relation. Finite Element simulation for the single back plate case has been done in ANSYS software. It is shown that Finite Element Method results have good agreement with analytical results. Also, a comparison between analytical and experimental results in the literature has been undertaken and it shows that the errors are negligible. Finally, the effect of geometrical parameters, such as diameter, thickness and the gap of the diaphragm on the Pull-in voltage, in both cases, has been investigated. It is shown that the increase of thickness or gap will increase the pull-in voltage, while the increase of diameter will decrease the value of Pull-in voltage. Also, it is shown that in all cases, the value of the Pull-in voltage in the dual back plate case is more than that in the similar single back plate.

Key Words: Micro circular plate, pull-in voltage, analytical solution.

TWO-OBJECTIVE OPTIMIZATION OF HEAT RECOVERY AND DESALINATED WATER PRODUCTION FROM A ONCE-THROUGH COOLING SYSTEM

A. K. Akbari
aliakbary29@yahoo.com
M. Gholinezhad
masoudgholinejad@gmail.com
O. Pourali(corresponding author)
pourali@kntu.ac.ir
M. Amidpour
amidpour@kntu.ac.ir
Dept. of Mechanical Engineering
K. N. Toosi University of Technology

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 137-147, Research Note

- Received 9 September 2014; received in revised form 3 March 2015; accepted 15 March 2015.

Abstract

A once-through cooling water system is a major source of water and energy loss in power plants located in coastal areas of the Persian Gulf; the relatively high temperature discharged water causes some operational problems in these power plants. In addition, the once-through cooling system is a major source of environmental pollution in power plants located along the south coastline of Iran. On the other hand, Persian Gulf coastal areas suffer from a shortage of natural fresh water; therefore, water treatment (natural fresh water production), and heat recovery from the discharged cooling water to the Persian Gulf is essential. In this research, heat recovery from the discharged cooling water to the Persian Gulf was studied, and application of the recovered energy for water treatment (fresh water production) using a multi-stage flash desalination system in thermal power plants was also investigated. For this purpose, a multi-stage flash desalination system was thermodynamically modeled, and then, a thermoeconomic model)based on exergy and economic analysis(was studied to determine the water cost. In the next step, a two-objective genetic algorithm was applied as a suitable optimization approach to obtain the optimal values of the decision variables (number of stages, top brine temperature, last stage temperature, terminal temperature difference of brine heater and condenser). Exergetic efficiency and the cost of water were considered the objective functions of the optimization procedure. By applying the optimization technique, the cost of water was minimized, while exergetic efficiency was maximized. In addition, the effects of heat recovery on the cost of the water desalination process were studied, and results demonstrated that the cost of water desalination was reduced by 13% using heat recovery from the discharged cooling water. Also, a sensitivity analysis was performed in order to show the relationship between the cost of water and some important parameters of desalination systems.

Key Words: Once-through cooling system, heat recovery, desalination, thermoeconomic analysis, Pareto front.

model length. The turbulence intensity across the tunnel working section was measured by the hot wire, and was obtained between 0.2 to 0.25%. To move the holder of the hot wire, a traversing system with three degrees of freedom was used. The pressure measurement of the flow field over the wing was also performed using a pitot tube for validation of the hot wire results. The experimental results showed multiple vortical flows over the wing. A dual vortex structure was formed above the wing surface before the leading edge crank, and another vortex flow was seen after the change of the leading edge sweep. The standard deviation of the measured velocity increased at the core of the vortices, reflecting the instability of the flow at that region. By increasing the angle of attack, i) the pressure drop increased and the vortices became wider; ii) the vortices moved inboard along the wing, and away from the surface; and iii) the flow separation initiated from the outer portion of the wing and progressed to its inner part. The vortices of the wing of the "sharp" leading edge were stronger than those of the "round" one.

Key Words: Lambda wing, hot wire, vortical flow, leading edge shape.

EFFECTIVE PARAMETER INVESTIGATION IN NUMERICAL SIMULATION OF CAVITATION AND NON-CAVITATION FLOW AROUND A DTMB 4119 STANDARD PROPELLER

A.R. Ayoobi(corresponding author)
ahmadreza.ayoobi@gmail.com
S. Kheradmand
kheradmand@mut-es.ac.ir
Dept. of Mechanical Engineering
Malek-Ashtar University

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 119-128, Original Article

- Received 1 October 2014; received in revised form 9 March 2015; accepted 4 April 2015.

Abstract
In this paper, cavitation and non-cavitation flows around a marine propeller, DTMB 4119, with the RANS method, the K- ŏ3b5 turbulence model, the Singhal transfer model and the mixed model, were solved. The velocity and pressure fields around the propeller surfaces were obtained using the equations of continuity, momentum and the K-ŏ3b5 model of turbulence. The pressure coefficient for two sections of the propeller with experimental data (Jessup 1989) in non-cavitation flow, and three sections in cavitation flow with numerical data (Sun 2008), was validated. Thrust, torque and efficiency coefficients were extracted for this propeller in eight simulations and compared with experimental data. The cavitation pattern was specified and also the position and the cavity development area were obtained. Furthermore, numerical investigations for the three important parameters, such as: advance coefficient, propeller working depth and surface roughness of propeller in cavitation flow, were undertaken. An advanced coefficient effect study on the propeller surfaces showed that with less advance coefficient, more vapor phase value is observed. Also, cavity boundaries are extended. Between advance coefficient values of 0.833 and 0.7, cavity volume results are less than the advance coefficient values of 0.7 and 0.6. So, the cavity boundaries are significantly extended. By downing the propeller working depth under open water conditions (increasing hydrostatic pressure), cavity boundary movement, cavity volume variation and curve peak reduction of the vapor phase volume fraction were investigated. The cavity center was away from the leading edge and the probability of tip vortex cavitation was reduced. The sustainability of tip vortex cavitation is more than sheet cavitation, the casue of which is, first, tip vortex cavitation, and then, sheet cavitation of the leading edge. Investigation on propeller surface roughness showed that optimal roughness height could be found. A and B models showed less cavity volume than the smooth model, while the C model showed more cavity volume. When roughness height was increased, the vapor phase volume fraction was firstly reduced, the results of A and B models being nearly the same. As a result, increasing the roughness height to a specific value caused a decrease in the vapor phase volume fraction value, which afterwards grew.

Key Words: Cavitation flow, propeller working depth, surface roughness height, cavity boundary.

ANALYTICAL SOLUTION FOR PULL-IN VOLTAGE OF A CIRCULAR MICRO-PLATE FOR SINGLE AND DUAL BACKPLATE CASES

A. Shooshtari(corresponding author)
shooshta@basu.ac.ir
M.Saadatmand
s.miladsaadatmand@gmail.com
Dept. of Mechanical Engineering
University of Bu Ali Sina

MECHANICAL BEHAVIOR OF POLYESTER RESIN AND ITS POLYMER CONCRETE UNDER STATIC AND DYNAMIC LOADING CONDITIONS

M. M. Shokrieh(corresponding author)
shokrieh@iust.ac.ir
S. Rezvani
sina_rezvani@iust.ac.ir
Dept. of Mechanical Engineering
Iran University of Science and Technology
R. Mosalmani
mosalmani@scu.ac.ir
Dept. of Mechanical Engineering
Shahid Chamran of Ahvaz University

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 97-105, Original Article

- Received 13 September 2014; received in revised form 31 January 2015; accepted 17 February 2015.

Abstract

Polymer concretes (PCs) have superior mechanical properties and performance in comparison with traditional cement concretes. However, reducing fabrication costs is a great concern for the increasing application of PCs. In this research, fine silica sand was used instead of the common usage of a mixture of large and small aggregates, in order to minimize the void content for dry packed aggregates. Moreover, polyester resin is used as the binder of the concretes, mainly because of its lower price in comparison with conventional epoxy systems. The specimens of polyester polymer concrete (PPC) were fabricated and tested based on standard testing methods. These tests include compression and flexural tests, and the results of the tests were compared with available experimental data for PCs fabricated with an epoxy resin system. Based on the results obtained, it is concluded that the proposed composition reduced fabrication costs and has acceptable mechanical properties. In addition, the strengthening ability of polyester resin in the introduced concrete was better utilized than the polymer concrete fabricated with epoxy resin. The mechanical behavior of the suggested PPC was studied under compression and bending loading conditions, and compared with that of the neat polyester resin. Next, the influence of the applied strain rate on the mechanical response of both neat polyester and PPC was investigated. Special specimens for testing PPCs under dynamic loading conditions were designed and fabricated. Multiple experiments were performed in the range of strain rates from 0.1 to 90 mm/s. The experimental results show that the mechanical behavior of polyester resin and PPC is extremely sensitive to the loading rate. By increasing the loading rate, the tensile strength of the neat polyester resin specimens was increased by nearly 41%, while the elastic modulus did not change. Also, the results show an increase up to 27% in the compressive strength of the PPC. Considering the strain rate dependent behavior of the neat resin, which is one of the main constituents of the PC, the strain rate dependent behavior of the PPC is justifiable. However, the strain rate sensitivity of the PPC is lower than that of the neat polyester resin.

Key Words: Polymer concrete, polyester resin, dynamic behavior, strain rate.

EXPERIMENTAL INVESTIGATION OF VORTICAL FLOW OVER A LAMBDA SHAPED WING WITH SHARP AND ROUND LEADING EDGES USING HOT WIRE ANEMOMETRY

M. D. Manshadi(corresponding author)
mdmanshadi@alum.sharif.edu
M. Eilbeigi
mehdi_eilbeigi@yahoo.com
M. Bazaz zadeh
bazazzadeh@mut-es.ac.ir
M. Kazem Sobhani
mohamadkazem_sobhani@yahoo.com
Dept. of Mechanical and Aerospace Engineering
Malek Ashtar University

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 107-117, Original Article

- Received 13 September 2014; received in revised form 14 February 2015; accepted 15 March 2015.

Abstract

In this research, the flow fields over two lambda shaped wings were investigated. The velocity distributions were obtained using a hot wire anemometer. A lambda wing is one type of delta wing with a crank in its trailing edge, and the application of this type of wing is increasing in flying wings. In the present work, two different leading edge shapes, namely, “sharp” and “round”, were applied for the lambda wing. The wing had two sweep angles of 55 and 30 degrees. The experiments were conducted in a closed circuit wind tunnel at V=20 m/s and angles of attack of 5 to 20, with the step of 5 degrees. The Reynolds number of the model was about2×10^5 according to the

S. M. Sadrameli(corresponding author)
sadramel@modares.ac.ir
Dept. of Chemical Engineering
Tarbiat Modares University

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 77-82, Original Article

- Received 17 May 2014; received in revised form 27 December 2014; accepted 17 January 2015.

Abstract

There are three factors which affect the efficiency of the solar panels: the semi conductor material, solar radiation flux intensity, and solar panel operating temperature. Increasing the elevated operating temperature in solar photovoltaic cells induces a significant decrease in the efficiency. For photovoltaic panels, high operating temperatures cause a drop in the conversion rate of about 0.3%-0.5% per Celsius degree over the nominal cell operating temperature of 25^0C. To keep the operating temperature lower, one can either improve the free cooling on the back of the panel, thanks to natural or forced convection, or try to absorb the excess heat by modifying the panel's architecture. The latter solution includes the use of phase change material PCM situated on the back of the solar panels. This paper represents the results of an experimental investigation into using PCM for thermal management of photovoltaic devices to maintain the temperature of the panel close to the optimum temperature. Polyethylene glycol 1000 (PEG 1000) was used as a PCM. The materials were injected in the cubic ducts, and these ducts were fixed on the back of the solar panel. Experimental runs were done in radiation of $800W/m^2$. Also, two angles of horizontal and 15 degree inclination were examined. Results show that the consumption of PCM improves the efficiency of solar panels in electricity production. Also, it can maintain the temperature of the panel 15^0C lower in comparison to a single panel without PCM. This causes an 8% improvement in overall efficiency. Also, an increase in panel inclination has a positive effect on cooling. This is because the gushing of liquid PCM makes a forced convection in the PCM ducts. Using phase change materials seems to be an effective method for the efficiency enhancement of solar cells.

Key Words: Free cooling, phase change material, photovoltaic, solar energy, temperature control.

NUMERICAL INVESTIGATION OF TWO-PHASE FLOW IN LOW-PRESSURE STAGES OF A STEAM TURBINE TO STUDY EFFECTS OF VARIATIONS IN CONDENSER PRESSURE

M.R. Vaghar
m.r.vaghar@ut.ac.ir
A. Nejat(corresponding author)
nejat@ut.ac.ir
School of Mechanical Engineering
University of Tehran

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 83-95, Original Article

- Received 17 may 2014; received in revised form 19 November 2014; accepted 8 December 2014.

Abstract

Modern steam turbines are used routinely in various power plant electricity productions. In order to obtain descriptions of the complete flow-field of the LP stages of a steam turbine, numerical simulation of a steam turbine is performed. In steam power plants, increasing condenser pressure leads to a complex three-dimensional two-phase flow. Any jump in the back pressure of the turbine outlet results in lower steam axial velocity at LP stages. Consequently, a stall region is formed in the last rotational blade of the LP stages, which causes unstable vibrations known as stall flutter. Numerical simulations are more robust in detailed examination of different phenomena in steam turbines, since experimental examination of the flow field is relatively expensive. In this study, the flow field of the whole LP region of a steam turbine (7 stages) is simulated with steady 3D computational fluid dynamics (ANSYS CFX). The flow path is defined by the ANSYS-Blade editor, and the computational 3D structured grid was generated by ANSYS-Turbo Gird software .In addition, all boundary conditions and initial conditions applied in this study are explained. The flow rate, pressure, temperature and mass friction on the blades are computed via CFD simulation, and overall, good agreement with the experimental data is observed. The efficiency, available work, streamline and pressure distribution of the LP region are reported for analysis of the turbine performance change, due to pressure, of the steam turbine outlet (Condenser Pressure) variations. Since any jump in the pressure of the turbine outlet results in lower steam axial velocity in the LP stages, the turbine performance deviates from its design point. In this study, a stall region, the alarm and critical pressure of a turbine outlet were calculated.

Key Words: LP stages of steam turbine, two phase steady flow, condenser pressure variations effect, 3D numerical simulation.

collaboration between scientists in the fields of engineering, humanities, computer, robotics, and psychology, a new science-research cross-disciplinary area of "Social Robotics" has evolved. Robots have been previously developed for various purposes and needs, and now we are exploring its application in "English language teaching and learning". With the proliferation of computers and mobile devices, Computer Assisted Language Learning (CALL) and Mobile Assisted Language Learning (MALL) have been in the limelight of second language instructional theories for about a decade or more. Robots not only have the features and interfaces already being employed in CALL and MALL, but are also capable of autonomous movement, visual and voice recognition, and physical and environmental interaction when equipped with various sensors. Additionally, robots are different from computers and mobile devices in that they have a friendly appearance and are capable of developing social relations. In this paper, the effect of a humanoid shaped robot as a teaching tool and/or assistant in English language classrooms in Iranian high schools is reported. The textbook used was the English book (Prospect-1) devised by the Iranian Ministry of Education for 7th graders, and the vocabulary taught and tested was taken from this book. Moreover, the treatment given by a teacher accompanied by a humanoid robot assistant in the RALL group took about 5 weeks, during which, half of the book was covered, and the non-RALL group was taught in a traditional method. This article covers an overview of the subject and our current activities/findings in the interdisciplinary field of Robotics Assisted Language Learning (RALL) in English as a Foreign Language (EFL) classes, and as a case study for the broader field of Social Robotics in Iran.

Key Words: Social robatics, RALL, 2nd language teching/ learning, educational technology.

OPTIMAL THRUSTER CONFIGURATION FOR A GEOSTATIONARY SATELLITE

A. Karmozdi
alikarmozdi@mech.sharif.ir
H. Nejat(corresponding author)
nejat@sharif.edu
Dept. of Mechanical. Engineering
Sharif University of Technology

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 65-76, Original Article

- Received 3 May 2013; received in revised form 1 December 2014; accepted 14 December 2014.

Abstract

Since the lifetime of a satellite is generally limited by the amount of fuel, reducing its fuel consumption is especially important. To reduce geostationary satellite fuel consumption, it is necessary to select a thruster configuration, which, in addition to satisfying all attitude control system (ACS) requirements (such as producing torques in all directions, and modifying the orbital inclination and radius offsets), minimizes fuel consumption during the liquid apogee motor (LAM) maneuver. In this paper, after dynamic modeling, simulation of the satellite control is undertaken in three control laws: the direction cosine matrix, Euler angles and quaternion. All simulations were performed using MATLAB. By comparing these three control schemes, the best control law is selected for use in simulation of the studied configurations. On the other hand, control with thrusters has some complexity because of their inherent characteristics, such as minimum pulse width, valve opening and closing delays, the ability to generate force only in one direction and the uncertainty in the nominal thrust. We have included the thruster specifications in the actuator model in our control simulations. Finally, after introduction of requirements for an acceptable configuration, by conducting several simulations, parameters such as fuel consumption and attitude control accuracy for a number of different configurations, consisting of eight or six thrusters parallel to the main axes of the satellite, have been computed and compared. In conclusion, configurations with minimum steady state error, with minimum fuel consumption and configurations that minimize fuel consumption and steady state error, together, have been introduced.The results of this paper can be used for determination of the optimum configuration of thrusters in design of the ACS system of geostationary satellites.

Key Words: Optimal thruster configuration, geosatellite ACDS, 3D body attitude control.

DESIGN AND SETUP OF A COOLING SYSTEM USING PHASE CHANGE MATERIALS (PCMS) FOR THE EFFICIENCY ENHANCEMENT OF SOLAR PANELS

S. Reza Mousavi
reza.mousavi1366@gmail.com

Key Words: Cylindrical shell, internal explosion, fragmentation, smooth particle hydrodynamic method.

PREDICTING LONGITUDINAL MOTION OF A PLANING CATAMARAN IN IRREGULAR WATERS

A. Najafi
najafi@mech.sharif.edu
M.S. Seif(corresponding author)
seif@sharif.edu
Dept. of Mechanical Engineering
Sharif University of Technology

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 41-55, Original Article

- Received 6 April 2014; received in revised form 12 July 2014; accepted 8 November 2014.

Abstract

High speed craft have longitudinal instabilities in waves and calm water, so, it is very important to know the dynamics and behavior of these vessels. In recent years, there has been more attention paid to the study of hull form optimization, movement improvement and instability control for planing marine vessels. The problem of increasing movement and acceleration of marine structures, especially in high speed craft, has a negative effect on the performance of the structure, its crew, passengers and equipment. So, estimation of vessel movement is an important part of its design. In this regard, wide studies have been carried out, experimentally and numerically, on the motion of these vessels in regular waters. But, in high speed craft, due to the complex and nonlinear behavior of the motion equation, the numerical solution of the equations of motion in the time domain is highly complex and time-consuming in calculating irregular waves. Therefore, researchers prefer to use experimental testing. Due to a limitation of equipment in marine laboratories, the use of irregular wave makers and expensive tests for real vessels in open seas, it is necessary to obtain an analytical solution for estimation of motion in irregular waves. The purpose of this study is to illustrate how we can achieve an analytical method, based on initial conditions, for planing vessels in irregular waters. In this study, an analytical model, with low computational cost, will be provided for the longitudinal movement of a catamaran. This model is based in consideration of analytical and numerical studies, and during this research, other published findings will be produced. The proposed method uses the principle of a superposition of forces, which has no antonym with the nonlinear motion of high speed craft. In this method, the hydrodynamic coefficients must be constant, relative to movement frequency, so, numerical evaluation of the hydrodynamic coefficients has been discussed in every motion frequency. Various experiments have shown the hydrodynamic coefficient tendency to be constant at high collision frequency. By examining the various hydrodynamic coefficients in collision frequencies, an increase in the frequency of collisions, as well as fixed coefficients, were observed. The method used in this research has acceptable answers for high wave lengths, with respect to vessel length.

Key Words: Hydrodynamic coefficients, irregular waves, Savitsky method, computational fluid dynamics, longitudinal motion.

IMPACT OF SOCIAL ROBOTS AS ASSISTANTS FOR ENGLISH LANGUAGE TEACHING IN IRANIAN SCHOOLS

M. Alemi(corresponding author)
alemi@sharif.edu
Dept. of Mechanical Engineering
Islamic Azad University, Tehran-west Branh
A. Meghdari
meghdari@sharif.edu
M. Ghazisaedy
m.ghazisaedy@yahoo.com
M. Zandvakili
mersi_zand@gmail.com
A. Karimian
ak_1371@yahoo.com
Dept. of Mechanical Engineering
Sharif University of Technology

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 57-64, Original Article

- Received 28 April 2014; received in revised form 17 November 2014; accepted 29 November 2014.

Abstract

For over 50 years, a great deal of effort has been expended on improving human efficiency by utilizing robots. Improvements in robotics technology have opened new possibilities for its effective use in social, cultural, therapeutic, and economic fields. As a result of successful

on the pumping action of the heart producing a pulsatile pressure gradient throughout the arterial system. The theory of microfluids exhibits the microscopic effects arising from the local structure, and the micro-motion of the fluid elements was developed. Such fluids support stress and body moments, including rotary inertia. There is a subclass of microfluids, namely, micropolar fluids, which support couple stress, body couples, microrotational effects and microrotational inertia. The micropolar fluid, e.g. liquid crystals, suspensions and animal blood etc., consists of randomly oriented bar-like elements or dumbbell molecules, and each volume element has a microrotation about its centroid, described in an average sense by the skew-symmetric gyration tensor. From a continuum point of view, the classical Navier-Stokes equations are incapable of explaining the theory of micropolar fluid as they contain no proper mechanism to account for the cellular microrotations. In this paper, an unsteady pulsatile laminar blood flow through a viscoelastic artery with large displacement and Cosserat continuum assumption has been developed and numerically investigated, where the blood was assumed to be a micropolar fluid. A finite difference Cosserat formulation is developed within the principles of continuum mechanics. Fluid flow simulation has been undertaken in different states, like a rigid flexible wall, together with classical theory. By comparing experimental data with Cosserat theory results, some unknown coefficients have been determined. The pressure and velocities of unsteady pulsatile blood flow have been obtained according to these coefficients by using a pressure correction numerical solution approach for fluid and coupling with solid equations. An arbitrary Lagrangian-Eulerian approach has been selected for fluid-structure interaction in this paper. The achieved results are in good agreement with experiment data and other analytical solution results. Results in this paper show that the micropolar fluid model of blood and the viscoelastic model of the artery, despite the existence of fluid solid interaction, increase, in accordance with the numerical results of valid experimental data.

Key Words: Cosserat continuous model, micropolar, fluid-structure interaction, viscoelastic artery, pulsatile blood flow.

SIMULATION OF EXPANSION AND FRAGMENTATION OF A CYLINDRICAL SHELL SUBJECTED TO INTERNAL DETONATION BY THE SMOOTHED PARTICLE HYDRODYNAMICS METHOD

M. V. Mousavi(corresponding author)
mvmousavi@mail.kntu.ac.ir
J. Zamani
zamani@kntu.ac.ir
Dept. of Mechanical Engineering
K. N. Toosi University of Technology

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 29-39, Original Article

- Received 20 February 2014; received in revised form 23 July 2014; accepted 6 December 2014.

Abstract

The elastic-viscoplastic behavior and failure of ductile metal shells under internal detonation have an application to military and scientific fields of research. It is an important issue in a variety of circumstances, like structure protection, weapon effectiveness and safety application. After starting detonation in a fully charged shell, pressure waves are generated and transferred to the shell wall. The amount of this pressure wave is several times greater than the ultimate strength of the material, and is close to the CJ pressure of the explosive.

Simulation of the process of fragmentation is very complex and challengeable. It involves many dynamic processes, such as detonation wave generation in explosives, interaction of the shock wave with the shell, expansion of the shell at a high strain rate, and eventually failure and fragmentation of the shell case. So, using a theoretical method is very sophisticated and, in some cases, impossible. Also, experimental testing is dangerous and requires expensive equipment to capture the dynamic properties of the shell. Therefore, identification and use of numerical methods that have the ability to realistically simulate this process have priority.

In this research, expansion, fragmentation and post failure behavior of OFHC copper cylindrical shells under internal detonation of a C4 charge, have been simulated by a smoothed particle hydrodynamic method (SPH). The main advantage of this method, in comparison to conventional methods, such as the element deletion method, is realistic simulation of the fragmentation process due to complete satisfaction of the mass conservation law in the problem domain. Spatial distribution and approximate number and velocity of the fragments are obtained by implementation of the SPH method. Also, the leakage of detonation through case fragment products was simulated. Comparison of the expansionary deformation profiles obtained from simulation with the experiments showed that the average difference is less than 8%. The fracture radius of the shell and the velocity of fragments are compared with theoretical relations and good agreement between them has been observed.

method, the Newmark method is used to solve the dynamic equations of the laminated composite plate. For active vibration control of the laminated plate, a feedback control algorithm is used. The model is applied for the solution of two illustrative cases and the results are presented and discussed.

Key Words: Plate vibration, mesh-less method, piezoelectric, sensor and actuator.

THERMOSYPHON ANALYSIS USING THE PARTICLE IMAGE VELOCIMETRY METHOD (PIV)

P. Hanafizadeh(corresponding author)
hanafizadeh@ut.ac.ir
M. R. Momenifar
m_momenifar@ut.ac.ir
Dept. of Mechanical Engineering
University of Tehran
M. Bastankhah
majid.bastankhah@epfl.ch
Dept. of Mechanical Engineering
Ecole Polytechnique Federal De Lausanne
M. H. Saidi
saman@sharif.ir
Dept. of Mechanical Engineering
Sharif University of Technolog

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 11-20, Original Article

- Received 25 November 2013; received in revised form 13 April 2014; accepted 21 June 2014.

Abstract

Thermosyphons are devices used to transfer heat from a hot to a cold source benefiting the effect of gravity. They consist of three main parts, namely, evaporator, condenser and adiabatic section. Working fluid absorbs heat from the heat source and delivers it to the condenser section and releases it into the environment. Thermosyphones are able to transfer heat between the heat sources and sinks. Due to the high latent heat of the working fluid, thermosyphones can transfer huge amounts of energy. Therefore, they are considered one of the best heat transfer devices. They are widely used in various industrial fields, such as in solar systems, microelectronic devices, CPU cooling and air conditioning. Most researchers in this field focus only on heat transfer characteristics, and, due to practical considerations, rarely consider their hydrodynamic specifications. Indeed, the hydrodynamic characteristics of thermosyphones considerably affect their performance. In this paper, the velocity profile in the liquid phase is determined via a particle image velocimetry technique (PIV). For this purpose, a typical thermosyphon has been designed and constructed with transparent up riser and down-comer sections. In this study, a circular thermosyphon is analyzed and water is used as a working fluid in the circular thermosyphon. At the beginning, the velocity field of the liquid phase is detected in the transparent thermosyphon using a high speed camera and an image processing technique. Subsequently, these pictures are used to generate the velocity profiles and are combined with theoretical analyses to evaluate the performance of the thermosyphon. The results are compared with numerical investigations and show good consistency. The results indicate that the particle image velocimetry (PIV) truly determines the hydrodynamic and thermal characteristics of the thermosyphones. Moreover, in this study, the effect of input heat and the inclination angle of the thermosyphon are investigated numerically. It has been shown that the maximum efficiency of thermosyphon is in a horizontal position.

Key Words: Heat pipe, thermosyphon, PIV, velocity profile.

NUMERICAL SIMULATION OF PULSATILE MICROPOLAR BLOOD FLOW IN A VISCOELASTIC ARTERY AND COMPARISON WITH RIGID AND ELASTIC ARTERIES

M. Mahmoodi
mehdymahmoody@gmail.com
M. Gorji(corresponding author)
gorji@nit.ac.ir
Dept. of Mechanical Engineering
Babol University of Technology
F. Boustani
f.boustani@gmail.com
Dept. of Mechanical Engineering
Iran University of Science & Technology

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 21-28, Original Article

- Received 16 December 2013; received in revised form 7 December 2014; accepted 14 December 2014.

Abstract

Under normal physiological conditions, the transport of blood in the human circulatory system depends entirely

Abstracts of Papers in English

ACTIVE VIBRATION CONTROL OF LAMINATED PLATE USING PIEZOELECTRIC SENSORS AND ACTUATORS

A. Bagheri
bagheri@guilan.ac.ir
A. Sadri(corresponding author)
amin.sadri.ac@gmail.com
J. Javadi Moghaddam
jalaljavadimoghaddam@gmail.com
Dept. of Mechanical Engineering
University of Guilan

Sharif Mechanical Engineering Journal
Volume 32, Issue 1, Page 3-19, Original Article

- Received 19 June 2013; received in revised form 1 November 2014; accepted 25 November 2014.

Abstract

Based on the desirable properties of smart materials, many studies on the application of these materials in the field of vibration control, damage detection, shape control and buckling control have been undertaken. For the design and analysis of smart beams, plates or shells, several considerations, such as the type and number of sensors and actuators and their locations, the method of modelling, and different analyses and the types of control system, are considered.

In this paper, vibration control of a laminated composite plate, with piezoelectric actuators and sensors, is investigated. Governing equations of the laminated composite plate were derived using piezoelectric constitutive equations and Hamilton's principle. For modelling of the laminated plate, one of the numerical mesh-less methods; Element Free Galerkin (EFG), is used. In such methods, for discretization of a continuous field, a scattered set of particles instead of elements are distributed in the field, and the method of moving least squares (MLS) is used to construct the shape functions. In order to model the displacement and strain field of the laminated plate, the first-order shear deformation theory (FSDT) is used to consider its shear deformation effects. To verify the results, the natural frequency of isotropic and orthotropic plates with different boundary conditions was calculated and compared with other studies. The accuracy of the piezoelectric modelling is verified by calculating the displacement in different points of a bimorph piezoelectric beam in response to applying the electrical potential. After verification of the proposed