by standard input data and compared with previous works, data. Results for a T-shaped mixer are in good agreement with reference data. Then, simulation of flow around stationary cylinders is done and results are compared with different rotating cases. It can be admitted that two rotating cylinders are better. However, two cases in which the rotational direction of cylinders are the same, improve mixing much better. In these two cases, cylinders force the fluid particles to enter the other fluids region and mix with each other.

Key Words: Micro mixer, active method, concentration, rotary cylinder, numerical simulation, finite element method.

Gh. Faraji(corresponding author)
ghfaraji@ut.ac.ir
Dept. of Mechanical Engineering
University of Tehran
J. Shahbazi Karami
Shahbazi.mech@gmail.com
Dept. of Mechanical Engineering
Shahid Rajaee Teacher Training University
S. Sadeghian
Samin.sadeghian@ut.ac.ir
Dept. of Mechanical Engineering
University of Tehran
DOI:10.24200/J40.2020.54028.1522

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 121-130, Original Article

- Received 14 October 2019; received in revised form 1 January 2020; accepted 7 January 2020.

Abstract
In this paper, the strain area has been designed as a diamond in a new Rubber Pad Tube Straining (RPTS) method, and the tests are applied to two layer copper-aluminum. Tubes used in this experiment are pure copper with dimensions with an outer diameter of 22 mm and a thickness of 0.85 mm and also for the aluminum tube grade 1100 with an outer diameter of 22 mm and a thickness of 1 mm, which, after cutting at a length of 110 mm in diameter, were placed in the die. To get the desired result, it was decided to use high-strength polyurethane. A mandrel, which has a convex portion in the form of a diamond, is pushed into the tube, while the tube is fixed by a cylindrical rubber. As a result, the initial diameter of the tube is increased locally, and then, when the mandrel goes down, the energy stored in the rubber returns the tube to its original dimensions. The simulation of the process using the Abaqus finite element software showed that the highest strain is obtained when the maximum diameter of the convex section is as large as the outer diameter of the tube. The strength of the two-layer tube in the third pass was 180/6 MPa, which grew to the same as the aluminum base metal and was lower than that of the copper base metal. The hardness of the copper section of two layer tube under the process compare to its base metal obtained from 92 HV to 130 HV. This condition had a growth for the aluminous layer of 40 HV in the base metal to 50 HV in the two-layer aluminum section. The simulation shows that regardless of the copper-aluminum-metal bonding region, the proposed method has major advantages including high strain homogeneity. The process force is also calculated in the first pass of about 31 tons.

Key Words: Severe plastic deformation, bi-metal tubes, finite element method, mechanical properties.

NUMERICAL INVESTIGATION ON INCREASING THE MIXING INDEX IN AN ACTIVE MICRO MIXER BY TWO ROTATING CYLINDERS

S. Mohammad Bagery
saeed_bagery@yahoo.com
M. Vajdi(corresponding author)
vajdi@uma.ac.ir
Dept. of Mechanical Engineering
University Mohaghegh Ardabili
DOI:10.24200/J40.2019.52118.1488

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 131-137, Research Note

- Received 1 December 2018; received in revised form 5 June 2019; accepted 2 July 2019.

Abstract
Micro-mixers are one of the important components of micro-fluid systems. Micro mixers are small devices that are used to mix two different fluids which can be liquid or gas phases. Since the scale of these devices is very small and fluid flows with very low Reynolds. At lower Reynolds numbers, the fluid's diffusivity is the dominant parameter to mix the fluids. If the flow is forced to create vortices, mixing phenomena will improve considerably. So using the appropriate geometries to create vortices and accelerate the mixing process along the channel is essential. In this study, two cylinders are used to enhance the mixing which is a passive method.
Despite other mixing accelerators, cylinders are better because of their smaller drag coefficient. Rotating cylinders can help the mixing phenomena by means of fluid no-slip boundary condition. Since a rotating cylinder can change the path lines of fluid particles and help the mixing process, both cylinders is used geometry rotate clockwise or counter clockwise. Simulation of mixing and fluid flow, accomplished with numerical methods and finite element. Governing equations are solved by means of COMSOL software and the results are shown as contours of pressure and concentration.
Four different cases are considered base on the combination of rotation direction of cylinders. To verify the numerical method, simulation of a T-mixer is done

estimated in the MATLAB software environment by applying the equations of motion governing an ellipsoidal body shape. It should be noted that, for deriving the remaining hydrodynamic coefficients of the mentioned prolate spheroid using HEKF, positions and velocities of the object in each time step are needed. So, the dynamic motion of the vehicle in the fluid is simulated using the transient tool of the ANSYS CFX software while benefiting from the remeshing algorithm. Furthermore, the hybrid form of the Extended Kalman Filter (EKF) is chosen as the equations governing the 3DoF motion of the ellipsoid is continuous and the measurements are discrete. Results obtained from the proposed method indicate a good agreement for estimated added mass derivatives in comparison with the available analytical ones. The following study is an effort made to introduce an innovative strategy to investigate the hydrodynamic coefficients of submersible platforms like submarines, torpedoes and AUVs.

Key Words: Hydrodynamic coefficients, kalman filter, computational fluid dynamics, prolate spheroid, dynamic mesh.

AN ANALYTICAL SOLUTION TO TWO-DIMENSIONAL UNSTEADY MASS TRANSFER EQUATION WITH ARBITRARY SOURCE TERM IN THE RIVER

N. Mashhadgarme
n.mashhadgarme@modares.ac.ir
M. Mazaheri(corresponding author)
m.mazaheri@modares.ac.ir
J. MohammadVali Samani
samani_j@modares.ac.ir
Dept. of Water Structures
Faculty of Agriculture
Tarbiat Modares University
DOI:10.24200/J40.2019.54224.1527

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 109-119, Original Article

- Received 32 September 2019; received in revised form 3 November 2019; accepted 9 December 2019.

Abstract
The concentration distribution in the water resources is analyzed with Mass transfer equation. Analytical solutions of this equation play an important role in understanding the mass transfer problem, mass transport parameter estimation, and numerical model verification. One of the powerful methods in solving nonhomogeneous partial differential equations analytically in one or multi-dimensional domains is Generalized Integral Transform Technique. This method is based on eigenvalue problem and integral transform that converts the main partial differential equation to a system of Ordinary Differential Equation (ODE). In this research, an analytical solution to two-dimensional Mass transfer equation with arbitrary emission time patterns of point sources was obtained at the finite domain in the open channels using Generalized Integral Transform Technique. In order to evaluate the extracted solution, the result of the proposed solution was compared with the Green's function method solution in the form of two hypothetical examples. In the first example, one point source with the compound exponential function was considered. In the second example, two point source with irregular time pattern at the distance from each other was assumed. The final results were represented in the form of the concentration contours at different times. The results show the conformity of the proposed solution and Green's function solution and report the suitable performance of the proposed solution. The presented solutions have various applications; they can be used instead of numerical models for constant- parameters conditions. The analytical solution is as an exact, fast, simple and flexible tool that is conveniently stable for all conditions; using this method, difficulties associated to numerical methods, such as stability, accuracy, numerical dispersion, etc., are not involved. The extracted solution in this research can adopt multiple pollutant sources with arbitrary time patterns and can be used as a benchmark solution for the numerical solution validation in two-dimensional mode.

Key Words: Mass transfer equation, point sources, arbitrary time pattern, generalized integral transform technique, finite domain.

EXPERIMENTAL INVESTIGATION AND NUMERICAL ANALYSIS OF NOVEL PROCESS TO PRODUCE OF HIGH STRENGTH BIMETAL TUBES BY RUBBER PAD TUBE STRAINING (RPTS) METHOD

O. Shapourgan
shapurgan@ut.ac.ir

F. Sadeghian Jahromi
s.farid1367@gmail.com
M. Beheshti(corresponding author)
m.behshti@eng.ui.ac.ir
Dept. of Chemical Engineereing
Esfahan University
DOI:10.24200/J40.2019.52540.1498

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 89-98, Original Article

- Received 29 April 2019; received in revised form 16 June 2019; accepted 26 June 2019.

Abstract
The purpose of this manyscript includes providing new analysis and optimization method in process systems which is achieved due to the strengths of the simultaneous analytical methods of energy-exergy for thermal and chemical systems. In this treatise, it has been tried to with doing a comprehensive research effort, a new method is proposed to target energy-exergy process systems with help of other techniques in order to be useful in analyzing and optimizing process systems. In addition, exergy, exergyeconomic and pinch methods have been introduced according to the need and importance of relevant analyzes. In the following, the developed method in the synchronous combination of Pinch-Exergy called Bridge-Exergy is presented at two different levels for analyzing the energy and exergy process systems simultaneously. The output of this developed bridge-exergy method is a new and correlated equation for the simultaneous analysis of energy and exergy, which it leads to creation of an Exergy Transmission Curve (ETC) for thermal equipment and an energy — exergy destruction curve (EDL) and an energy — exergy cost destruction curve (ECDL) for equipment including pressure and composition change. Hereafter using this method of combining and analyzing the equipment of the process system, by using the artificial intelligence method called differential evolution algorithm, the optimization and integration of the whole process system are done simultaneously. Finally, based on the developed energy-exergy analysis method, a new algorithm is introduced and evaluated for the design of optimal process systems. As a field study for investigating the bridge-exergy combination method and optimization algorithm, NGL production process has been briefly investigated and thoroughly analyzed and optimized. In the process of natural gas separation, using the bridge-exergy combination method and artificial intelligence method, the differential evolution algorithm, the operating cost is reduced about 44 % and the annual profit is increased to about 814400 ($/year).

Key Words: Integration, bridge -exergy analysis (ETC), energy -exergy destruction curve (EDL), energy -exergy cost destruction curve (ECDL), optimization.

HYDRODYNAMIC COEFFICIENTS ESTIMATION OF A PROLATE SPHEROID USING COMBINATION OF HYBRID EXTENDED KALMAN FILTER (HEKF) AND COMPUTATIONAL FLUID DYNAMICS (CFD)

M. Rasekh
m.rasekh@sru.ac.ir
M. Mahdi(corresponding author)
m.mahdi@sru.ac.ir
Dept. of Mechanical Engineering
Shahid Rajaee Teacher Training University
DOI:10.24200/J40.2019.53476.1515

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 99-108, Original Article

- Received 21 MAy 2019; received in revised form 18 October 2019; accepted 26 October 2019.

Abstract
In the present study, a novel method has been introduced to estimate the hydrodynamic coefficients of an axisymmetric submersible prolate spheroid. In this heuristic procedure which is based on combination of the nonlinear Hybrid Extended Kalman Filter (HEKF) observer and Computational Fluid Dynamics (CFD), all of the hydrodynamic derivatives of a prolate spheroid in horizontal plane are estimated to simulate the Three Degrees of Freedom (3DoF) equations of motion. For this purpose, first, a CFD-based numerical model has been developed for an axisymmetric prolate spheroid by the commercial code ANSYS CFX software using the remeshing algorithm in dynamic mesh method. In this algorithm, the grid around objects deform locally using the Arbitrary Lagrangian-Eulerian (ALE) form of the governing fluid equations. When this deformation reduces the mesh resolution, significantly, the grid is remeshes and improved. By performing the corresponding calculations along with the simulation, some hydrodynamic derivatives were obtained. Then, utilizing the hybrid kalman filter simulation code based on the parameter identification, other unknown derivatives were

a nonlinear filter. To identify model changes after fin failure, the aerodynamics of body and control surfaces are modeled separately. A percentage of the fins which have the ability to generate aerodynamic force is modeled by a parameter and these parameters are estimated over time, using a nonlinear estimator. Parameters estimation through the filtering approach is an indirect procedure, consisting of transforming the problem into a state estimation problem. The value of these parameters will be between 0 and 1. Where value 1 corresponds to the health of the fin and value denotes whether the fin has not opened or has been destroyed completely. The proposed method detects and isolates fins damage in a few seconds with good accuracy. Simulation results show that the new failure diagnosis algorithm estimated fins health percentage with good accuracy. After estimating the parameters, they can be sent to the controller and the controller has changed control signal.

Key Words: Fin failure detection, nonlinear kalman filter, fin control agile vehicle, parameter estimation, aerodynamic coefficient estimation.

EFFECT OF MULTIPHASE LATTICE BOLTZMANN COLLISION MODELS WITH SINGLE- AND MULTI-RELAXATION TIMES FOR SIMULATION OF LIQUID-VAPOR TWO-PHASE FLOWS USING TWO DIFFERENT FORCING SCHEMES

E. Ezzatneshan(corresponding author)
e_ezzatneshan@sbu.ac.ir
R. Goharimehr
r.goharimehr@mail.sbu.ac.ir
Dept. of New Technologies Engineering
Shahid Beheshti University
DOI:10.24200/J40.2019.52912.1505

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 77-88, Original Article

- Received 16 March 2019; received in revised form 1 June 2019; accepted 7 July 2019.

Abstract
In this paper, the multiphase lattice Boltzmann collision models are evaluated by a comparative study for the simulation of liquid-vapor two-phase flow problems. Herein, the single-relaxation-time (SRT) scheme based on the Bhatnagar-Gross-Krook (BGK) approximation and the multiple-relaxation-time (MRT) method with two different forcing schemes are considered. The pseudo-potential Shan-Chen (SC) model is used to resolve the inter-particle interactions between the liquid and gas phases. In the standard form of the SC model, the interaction force is imposed in the momentum field which unphysically causes the density ratio to change with the variation of relaxation time. In this study, a modified form of this model is implemented to decouple these two physical parameters. Herein, the interaction force is imposed using the exact difference method (EDM). The efficiency and accuracy of the present numerical scheme based on the lattice Boltzmann method (LBM) with the SRT and MRT schemes are examined for simulation of two-phase flows in different conditions. The equilibrium state of a droplet in the periodic flow domain and on the flat surface with hydrophobic and hydrophilic wetting condition are computed to investigate the robustness and performance of the collision operators applied. The results obtained for these problems are compared with the analytical solutions which shows a good agreement. The collision of a droplet on the liquid film at various flow conditions is investigated and the predicted results are presented at a range of the Weber and Reynolds numbers. The present study demonstrates that the SRT model suffers from the spurious velocity in the interfacial region which causes numerical instabilities at moderate Reynolds and Weber numbers. It is found that the MRT model is stable for all the cases considered in the present work even at high Reynolds and Weber numbers. In terms of the computational efficiency, the SRT scheme is slightly attractive, although the computational cost of this model is not considerably lower than MRT scheme. The present study suggests the lattice Boltzmann method with the MRT collision operator incorporated with the EDM technique is robust, sufficiently accurate and computationally efficient to resolve the practical liquid-vapor two-phase flow structures and properties.

Key Words: Multiphase lattice boltzmann method, shan-chen model, multi-relaxation time, exact difference method.

ANALYSIS AND OPTIMIZATION OF THE NGL PLANT USING THE EXTENDED BRIDGE - EXERGY METHOD AND DIFFERENTIAL EVOLUTION ALGORITHM

heat transfer deterioration phenomena is conducted utilizing the conjugate heat transfer simulations results of methane inside the MTP channel for heat transfer rate, inlet pressure and temperature, inlet mass flux, and different surface roughness. Further, a few relations are derived for predicting the onset of methane heat transfer deterioration along the rectangular cooling channel in a range of pressure from 6 MPa to 20 MPa. The relative error of derived relations with numerical data is less than 1 percent. Finally, some methods for controlling the heat transfer deterioration phenomena have been presented.

Key Words: Conjugate heat transfer, trans-critical methane, pseudo-critical temperature, rectangular channel, heat transfer deterioration.

INVESTIGATION THE EFFECTS OF LEAN IN A CENTRIFUGAL PUMP IMPELLER ON ITS AERODYNAMIC PERFORMANCE

A. Asgarshamsi(corresponding author)
a_shamsi@gut.ac.ir
M. Rezazadeh
rezazadeh@gut.ac.ir
A. Iravani
iravani.arash44@gmail.com
Dept. of Mechanical Engineering
Golpayegan University of Technology
DOI:10.24200/J40.2019.52549.1500

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 57-65, Original Article

- Received 13 March 2019; received in revised form 7 September 2019; accepted 13 October 2019.

Abstract
Centrifugal pumps are used in the various industries as one of the important components responsible for the movement of the incompressible fluids. In this paper, the fluid flow of water in a centrifugal pump is simulated with a three-dimensional numerical solution. The operating fluid is considered to be incompressible and in steady state condition. The numerical solution is performed using a ANSYSY 16.0 software. The provided results are validated by the pump manufacturer's characteristic curves. One of the effective factors in increasing the efficiency of the centrifugal pump impeller is the change in radial distribution of surface centers of two-dimensional profiles. In this research, these changes have been done in tangential direction. The change of the lean angle in two modes of fixed hub and fixed shroud has been performed. The simulation of the flow field for various lean angles showed that the creation of a negative lean angle has been increased the efficency by 0.51% for the fixed hub and 1.34% for the fixed shroud.

Key Words: Efficiency, centrifugal pump, impeller, numerical solution, lean angle.

AGILE FLIGHT VEHICLE FIN FAILURE TOLERANT CONTROL USING INERTIAL MEASUREMENT UNIT DATA

A. Ashrafifar
ashrafimut@gmail.com
M. Fathi Jegarkandi(corresponding author)
fathi@sharif.edu
Dept. of Aerospace Engineering
Sharif University of Technology
DOI:10.24200/J40.2019.52922.1506

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 67-75, Original Article

- Received 16 March 2019; received in revised form 3 August 2019; accepted 25 August 2019.

Abstract
In this paper, a new model-based fault detection method for an agile supersonic flight vehicle is presented. A nonlinear model, controlled by a classical closed loop controller and proportional navigation guidance in interception scenario, describes the behavior of the vehicle. The proposed FDD method uses Inertial measurement unit (IMU) data and nonlinear dynamic model of the vehicle to inform fins damage to the controller before leading to an undesired performance or mission failure. Broken, burnt or not opening of control surfaces causes a drastic change in aerodynamic coefficients and thus dynamic model. Therefore, in addition to the changes in the control forces and moments, system dynamics will change too, leading to the failure detection process being encountered with difficulty. To this purpose, an equivalent aerodynamic model is proposed to represent the dynamics of the vehicle, and the health of each fin is monitored by the value of a parameter which is estimated using

YB:YAG FIBER LASERS UNDER END-PUMPING

E. Amiri
ehsanamiri.ea1369@gmail.com
M. Mojahedi(corresponding author)
mojahedi@gut.ac.ir
Dept. of Mechanical Engineering
Golpayegan University of Technology
DOI:10.24200/J40.2019.52516.1497

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 35-44, Original Article

- Received 8 January 2019; received in revised form 10 August 2019; accepted 7 August 2019.

Abstract

The solid state lasers consist of host mediums that are doped by an active ions. The host environment is built in the form of slab, rod, disk and fiber according to the geometric shape. In most cases, the solid lasers can be stimulated by diode pumping. During the pumping, some of the pumping energy at the crystal is converted to heat and this heat causes the temperature gradient in the crystal. The temperature gradient can produce thermal stresses and even lead to fracture in the crystals. In the fiber lasers, these phenomena disturb the laser operation and decrease the maximum input power of the laser. In this paper, the thermo-mechanical behavior of single clad Yb:YAG fiber laser is discussed under continuous end pumping. The single clad fibers have been modeled with single and four cores and they are simulated by finite element software (Ansys). Since the thermal loading and geometry of the fiber have symmetry planes, in the modeling, one quarter model is used to determine the temperature and stress distributions. In the Ansys software, the coupled thermal-structural analysis is utilized. First, the temperature distribution is calculated in the thermal environment of the software and then thermal stress fields for single core and four-core fiber lasers are obtained by applying the temperature distributions calculated through the thermal analysis. In the analysis, the mesh dependency is evaluated and for achieving convergence, the required number of element is determined. The temperature distributions in single and four core fibers, with two positions for cores, are calculated. For any case, the calculated temperatures are compared with each other and the position of core related to smaller temperature is selected. The results show that, in comparison to single-core fiber, a four core fiber has smaller maximum stress at the same power and higher pumping fracture power. Finally the other thermal effects of the pumping, including thermal lensing, for single and four core fiber are compared with each other and it is shown that the four core fiber laser has better performance in terms of thermomechanical output.

Key Words: Fiber laser, Yb:YAG crystal, four core, continuous pumping, thermal-structural analysis.

PREDICTION OF HEAT TRANSFER DETERIORATION ONSET OF TRANS-CRITICAL METHANE IN A RECTANGULAR CHANNEL

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/J40.2019.52922.1506

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 45-55, Original Article

- Received 24 February 2019; received in revised form 3 June 2019; accepted 13 July 2019.

Abstract

The coolant fluid inside the cooling channels may experience flow regime change and heat transfer deterioration. Heating methane at super-critical pressure results in a higher than pseudo-critical temperature and pseudo-phase change happens. Furthermore, in critical conditions, there is the possibility of heat transfer deterioration at high heat fluxes and low mass fluxes. In these areas, heat transfer from the wall to the fluid is disturbed due to the decrement in the heat transfer coefficient. Also, the wall temperature increases and there is the melting possibility of the engine wall. As a result, the study of this phenomenon is important for the thermal analysis of cryogenic methane for cooling applications. In the present article, a three-dimensional solver is developed for the simulation of the conjugate heat transfer inside a rectangular channel with trans-critical methane coolant. A finite volume scheme is utilized for the discretization of the governing equations. An iterative solution method based on the SIMPLEC algorithm is used to solve the equations. The solver is developed based on the thermodynamic and transport property relations corresponding to the coolant flow conditions in the trans-critical regime. The parametric study of the

Low-weight, low power consumption, no moving parts and flexibility of use are some of its advantages over passive control methods.
In this paper, a User Defined Function (UDF) code was hooked into the main flow solver code to model the momentum injected to the flow by DBD actuator. Among different methods that were proposed, Shyy numerical method is selected. The selection process is based on pros and cons of different first principle and phenomenological methods which were published in recent years.
In order to validate the results of the air flow in the presence of DBD actuator two test cases are selected, the flow around the Flat Plate and NACA 0015 airfoil. Different flow variables including the velocity profile and pressure distributions are obtained and compared to the reference data. 3D effects of DBD actuators are also investigated by using a NACA 0015 wing model. Two tests are carried out. First, the effect of doubling the DBD field is studied. After doubling the strength of DBD field, the control power of actuator enhanced. Pressure distribution in the midpoint of wing clarifies this fact. Second, the effect of span-wise DBD actuators is investigated. In three locations DBD roll installed on the wing platform and its effect on the flow variables carried out. This experiment clarified that the best location to use DBD plasma actuators would be the onset of flow separation. In this location, the lift to drag ratio of the wing is maximum.
Also we studied the DBD parameters which affect the Ionic wind strength, Frequency and Voltage of the power supply. Increasing voltage and frequency would make plasma more effective (this is also concluded from lift to drag ratio) but the trend is not linear for voltage.

Key Words: Active flow control, DBD plasma actuators, numerical simulation, separation control, Shyy method.

EXPERIMENTAL INVESTIGATION ON SPIN EFFECT UPON BURNING RATE OF ALUMINIZED COMPOSITE PROPELLANT

A.R. Mohammadi
mohama_a@yahoo.com
Institute of Space Transportation Systems
Iranian Space Research Center
M. Farahani(corresponding author)
mfarahani@sharif.edu
M. Goodarz
mgoodarz202@gmail.com
Dept. of Aerospace Engineering
Sharif University of Technology

DOI:10.24200/J40.2019.52415.1494

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 29-34, Original Article

- Received 7 January 2019; received in revised form 13 July 2019; accepted 17 September 2019.

Abstract
This research is conducted to study acceleration effect on the burning rate augmentation of an aluminized solid propellant based on HTPB as an effective factor determining combustion chamber pressure. A centrifugal experimental setup was designed to obtain a uniform acceleration field by rotating the test motor around its longitudinal axis. A cylindrical port propellant grain was used in the test motor which had an inner diameter of 30 mm, an outer diameter of 60 mm and a length of 52 mm, so acceleration vector was always perpendicular to inner burning surface of propellant. Inner radius of tube was small, so the magnitude of acceleration increased as the grain burned back. The pressure of combustion chamber was changed from 30 bar 80 bar by changing the nozzle throat and the magnitude of acceleration changed from 2g to 60g by changing the rotational speed of solid rocket motor. Pressure of combustion chamber was measured. An analytical 0D code was used to compute burning rate augmentation of propellant in acceleration field. Nozzle throat diameter is another parameter controlling the pressure field of combustion chamber of solid rocket motor. Thermomechanical erosion of nozzle throat is significant in aluminized propellant, so erosion of graphite throat insert was measured and taken into account in 0D code. Burning time of all tests was below 2 sec. due to small web of the grain. As investigated, at low acceleration level (below 5g), burning rate of propellant is not sensitive to acceleration while in the condition in which acceleration changed from 30g to 60g, burning rate augmentation ratio increased from 1 to 1.5. The transient behavior of burning rate augmentation in acceleration field was obvious in obtained results due to short burn time of SRM. At high acceleration level, inner cylindrical surface of inhibitor was coated with aluminum oxide particles.

Key Words: Acceleration, burning rate, solid rocket motor, composite propellant.

THERMO-MECHANICAL ANALYSIS OF SINGLE AND FOUR-CORE

efficients of the system compared to the main cylinder. Furthermore, the performance of the new configuration in reducing the drag force is better than those of each of the splitter plate and control cylinder alone. At the condition with C/D=4 and L/D=1 the drag and lift coefficients of the system are reduced about 57% and 63% respectively compared with those of the main cylinder.

Key Words: Turbulent flow around cylinder, passive flow control, vortex shedding, splitter plate, control cylinder.

AGING HEAT TREATMENT AND SPECIMEN'S GEOMETRY: FRACTURE BEHAVIOR OF AL2024-T8 SHEET UNDER TENSION

K. Pakravan
kiana.pakravan@yahoo.com
A. Hosseini Monazzah(corresponding author)
a_hosseinimonazzah@sbu.ac.ir
Dept. of Mechanical and Energy Engineering
Shahid Beheshti University
DOI:10.24200/J40.2019.51936.1483

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 11-18, Original Article

- Received 15 December 2018; received in revised form 18 May 2019; accepted 26 June 2019.

Abstract

The effect of artificial ageing on tensile properties and fracture behavior of 2024 Al alloy in the presence and absence of notch was investigated. The tensile samples having two V notches, as well as, common tensile specimens were prepared according to ASTM E8. T8 heat treatment consisting of solutionizing, cold rolling, and aging was selected to establish precipitation hardening in Al2024 alloy. Considering T8 condition, following solutionizing of Al sheet at 500^0C, samples were cold rolled with 30% reduction. Finally artificial aging was applied at 200^0C via different times. The results of Brinell test shows that the maximum value of hardness was achieved at 1 hour, which indicated the peak aging condition. Moreover, soaking samples in aging condition at 200^0C for 15 minutes and 4 hours were considered as under-aging and over-aging states, respectively. Similar to hardness results, the superior amount of yield strength and ultimate tensile strength of un-notched specimens were observed at peak aging time, while in notched samples, the role of aging time was not prominent due to the presence of V shaped flaws. By applying notches on specimens, the yield and ultimate stress were found to increase with simultaneously decrease in elongation at fracture. It was found that due to localizing plastic deformation around flaws, the presence of notch increases the strength of 2024 Al alloy, while elastic behavior controls deformability far from notches. The energy which was absorbed till fracture, was indicated as toughness and it was found to be absolutely dominated by the elongation at fracture and significantly decreased in notched specimens. The peak-ageing condition of notched tensile specimens had the lowest toughness compare to under-ageing and over-ageing conditions. Although three fracture mechanisms, such as matrix deformation, particle cracking and matrix/particle debonding observed in all samples, the distribution of these mechanisms were more homogeneous in un-notched specimens. In the other words, one may claim that the presence of flaws enhanced the cracked particles, which accompanied with lowering matrix deformability lessened the elongation and toughness.

Key Words: Aluminum 2024, t8 heat treatment, tensile behavior, fracture mechanisms.

INVESTIGATION OF DIFFERENT ASPECTS OF DBD PLASMA ACTUATORS ON AIRFLOW AROUND A WING

S. Kabirian(corresponding author)
Siavash.Kabirian@aut.ac.ir
A.R. Jahangirian
ajahan@aut.ac.ir
Dept. of Aerospace Engineering faculity
Amirkabir university of technology
DOI:10.24200/J40.2019.52342.1493

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 19-28, Original Article

- Received 5 January 2019; received in revised form 17 July 2019; accepted 24 July 2019.

Abstract

Active control of airflow by using Dielectric-Barrier-Discharge (DBD) is a recent development in flow control theory.

Abstracts of Papers in English

PASSIVE CONTROL OF TURBULENT FLOW AROUND A CYLINDER USING COMBINATION OF SPLITTER PLATE AND CONTROL CYLINDER

S.A.R. Erfanian
rerfani92@gmail.com
A.R. Jahangirian(corresponding author)
ajahan@aut.ac.ir
Dept. of Aerospace Engineering faculity
Amirkabir university of technology
DOI:10.24200/J40.2019.52091.1485

Sharif Mechanical Engineering Journal
Volume 36, Issue 1, Page 3-10, Original Article

- Received 2 December 2018; received in revised form 12 June 2019; accepted 18 June 2019.

Abstract
The turbulent flow around a circular cylinder at the Reynolds number of 13100 is controlled using the combination of downstream splitter plate and upstream control cylinder as a new configuration. For this purpose, first a splitter plate with the length of L is attached to the downstream base of the cylinder and its length is varied to yield the optimum plate length in which the forces are minimum. The Reynolds averaged Navier Stokes equations is solved by a finite volume method together with a K-w SST turbulence model. Results show that the presence of the splitter plate has great impact on the flow characteristics and reduces the time average of the drag coefficient and amplitude of the lift coefficient oscillations. At L/D=1 the drag and lift coefficients of the system is reduced about 48% and 97% respectively. In the next part, a control cylinder with the smaller diameter than the main cylinder is placed at the upstream and the center to center ratio is varied. Results show that the control cylinder reduces the system drag coefficient in all the investigated distances while reduces the lift coefficient just in some bigger distances. At C/D=4 the drag and lift coefficients of the system is reduced about 35% and 11% respectively. In the last part, the simultaneous combination of upstream control cylinder and downstream splitter plate in their optimum configurations is used. Results show that this configuration significantly reduces the drag and lift co-