Abstract

Particle separation has received much attention in recent years due to its widespread use in various sciences such as engineering, medicine, and biotechnology. Separation of blood cells, detection of cancer cells, separation, and manipulation of particles of different sizes are among the applications of particle separation in the biomedical field. Lab-On-a-Chip (LOC), micro-total analytic systems (μTAS), and point-of-care diagnostics (POC) are some applications of microfluidic devices. Hence, microfluidics is an interdisciplinary area and has numerous applications in biomedical, chemistry, medicine, disease diagnostics, electronics industry, etc. The use of microfluidic devices has been the focus of attention in the last few decades. These devices have many advantages such as high efficiency, low cost, and environmental compatibility. The unique features of microfluidic devices have led to the use of a variety of techniques for rapid separation of particles with high efficiency. Many progresses have been made over the last two decades in particle separation. The performance of the devices used for separation of particles is evaluated according to the separation time, separation efficiency, and its applications. High separation accuracy can be achieved in continuous microfluidic devices since the volume of fluid in microfluidic devices is very low and their use is characterized by several advantages including low cost and short analysis time. In general, particle separation is performed using both active and passive methods. In the active methods, an external force is used as the driving force and in the passive ones, the particle separation is based on the geometry of the device without the use of external force. Active separation is carried out using magnetic, electrical, acoustic, optic and thermal forces. Passive separation methods for particle size-based separation, filtration, Zweifach-Fung effect, inertia, and Dean vortex as well as microwave separation. In the present work, the active and passive separation methods are described and the governing equations (small-scale flow) and necessary assumptions are considered. Also, the methods of fabrication of microfluidic devices are discussed.

Key Words: Particle separation, active methods, passive methods, numerical simulations, experimental methods.

SENSITIVITY ANALYSIS OF THE DOMINANT DESIGN PARAMETERS OF SUPERSONIC SEPARATORS FOR GAS COMPOSITIONS

K. Ghorbanian(corresponding author)
ghorbanian@sharif.edu

M.R. Amini Magham
m_amini@ae.sharif.edu
Dept. of Mechanical Engineering
Sharif University of Technology
DOI:10.24200/J40.2021.55595.1549

Sharif Mechanical Engineering Journal
Volume 37, Issue 1, Page 111-123, Research Note

- Received 28 May 2020; received in revised form 15 November 2020; accepted 3 January 2021.

Abstract

Separation of mixtures, gases, and phases is of great industrial importance. Various methods and equipment are utilized for the separation. One of the most novel and innovative as well as efficient methods is supersonic separation where acceleration of the flow mixture is accompanied by fast reduction in temperature and pressure. As a consequence, heavier components which have a higher condensation temperature will liquefy faster than lighter components. Swirling flow and centrifugal force will drive heavier condensed droplets toward the outer wall and separate them from lighter components in the gas phase. In general, supersonic separators consist of a converging-diverging nozzle and a cyclone separator. Due to no moving or rotating parts, supersonic separators have high reliability, long endurance, easy manufacturing, and less expensive operation and maintenance. In this paper, the one-dimensional design of a supersonic separator for separation and purifying of Methane from natural gas mixture is presented. Since separation process is in vicinity of saturation line, real gas model from NIST REFPROP software is used to estimate the fluid thermodynamic properties. For the design of cyclone separator and phase separations, instantaneous nucleation at the nozzle discharge is assumed and displacement of smallest droplet (as the worst case) is only considered. After calculating the supersonic separator geometry, performance sensitivity to changes in inlet temperature, pressure, and gas composition is studied, the effectiveness of these parameters is compared and rate of performance change is calculated. At next step, the simultaneous change of the inlet conditions is investigated and effect of each parameter on the effectiveness of others is studied. Finally, in order to optimize phase separation in various operating conditions, effects of swirler angle and swirl intensity on separation length are studied and optimum swirl intensity to gain the best possible separation performance with a fixed geometry separator for different inlet conditions is determined.

Key Words: Separation, supersonic, converging- diverging nozzle, cyclone separator, methane.

applied to the ejector walls. Then, the ejector geometry is optimized using Multi-Objective Genetic Algorithm (MOGA) in order to reach greater efficiency. Optimization is performed considering 4 geometric parameters including primary nozzle exit diameter, nozzle exit position, diameter, and length of the constant area section. Sensitivity analysis results show that the diameter of constant area section has major effect on the entertainment and pressure ratios as two objective functions. The nozzle exit position and external diameter of primary nozzle are the second and third dominant parameters that respectively influence the performance of the ejector, while the effect of constant area section length is negligible. Results indicate that by increasing the pressure ratio, the shock train moves upstream and at the design point, the last oblique shock is located in the exit of the constant area section, letting the remaining pressure recovery be done in the subsonic diffuser which reduces the pressure losses and increases the efficiency. Above the critical pressure ratio, due to the movement of the shock train to upstream weakening the shock strength, the suction pressure increases. Then, the pressure difference is reduced, leading to the lower secondary mass flow suction. The optimized ejector in the double choking condition has a 11.8% higher entertainment ratio and its operational range is enhanced by 5 percent in comparison to the original geometry.

Key Words: Single phase ejector, 2D simulation, multi-objective genetic algorithm, sensitivity analysis.

STUDY THE EFFECT OF SUCTION FLOW CONTROL ON THE AERODYNAMIC PERFORMANCE OF A WIND TURBINE USING 2D SECTION NUMERICAL RESULTS

M. Asghari
asghari.mahsa.92@gmail.com
S. Karimian(corresponding author)
karimian@modares.ac.ir
Dept. of Mechanical Engineering
Tarbiat Modares University
M.Y. Hashemi
m.y.hashemi@azaruniv.ac.ir
Dept. of Mechanical Engineering
Azarbaijan Shahid Madani University
DOI:10.24200/J40.2021.56336.1561

Sharif Mechanical Engineering Journal
Volume 37, Issue 1, Page 79-89, Original Article

- Received 31 August 2020; received in revised form 30 December 2020; accepted 9 January 2021.

Abstract

In the present work, the numerical investigation of a wind turbine airfoil with suction flow control is initially performed to find the relative improvement in its aerodynamic efficiency. Consequently the study of turbine performance is accomplished using 2D simulation results as extending the flow control to wind turbine blades. The latter is incorporated via definition of new input data file in Q-blade code including complete set of section aerodynamic coefficients for both with and without suction control against variety of angles of attack and velocities. The benchmark turbine in this study is NREL 5 MW. It is shown that using suction in 1/3 of chord length would lead in better enhancement. Simulation of the airflow with and without suction control has been presented in three different locations in the chord-wise direction (26%C, 33.5%C and 50%C) and for several intensities or suction speed ratios (0.022, 0.045, 0.11, and 0.155). The turbulence model is transitional 4-equation SST model. The aerodynamic improvement in section is dominant and this method could tolerate stall behavior. Wind turbine power and thrust coefficients were analyzed via Q-blade versus tip speed ratio and flow control parameter. The range of tip speed ratio studied here is 3.5 to 15. The results depicted that applying suction in certain part of the blades could increase output power at low tip speed ratio. Applying suction at 33.5% of chord length improves the average power coefficient by 4.1%.

Key Words: Flow control, suction actuator, wind turbine, power coefficient, aerodynamic coefficients.

COMPREHENSIVE STUDY OF PARTICLE SEPARATION IN MICROFLUIDIC DEVICES

M. Nazemi Ashani
msn.nazemiashani@gmail.com
M. Bayareh(corresponding author)
m.bayareh@sku.ac.ir
B. Ghasemi
ghasemi@sku.ac.ir
A. Shiriny
shiriny@stu.sku.ac.ir
Dept. of Mechanical Engineering
Shahrekord University
DOI:10.24200/J40.2020.55816.1552

Sharif Mechanical Engineering Journal
Volume 37, Issue 1, Page 91-110, Rview Article

- Received 1 July 2020; received in revised form 12 October 2020; accepted 18 October 2020.

the transmissibility decreases and its maximum value falls out of the operating frequency range, the damping ratio and dynamic stiffness are increased.

Key Words: Hydraulic engine mount, inertia track, decoupler, optimization.

INVESTIGATION OF TEXTURE INTENSITY EFFECT ON DEFORMATION BEHAVIOR OF EXTRUDED MG-0.8WT%Y ALLOY UNDER TENSILE AND COMPRESSIVE LOADING USING THE CRYSTALLINE PLASTICITY FINITE ELEMENT METHOD

A. Mirzakhani
a.mirzakhani@mech.sharif.edu
A. Assempour(corresponding author)
assem@sharif.edu
Dept. of Mechanical Engineering
Sharif University of Technology
DOI:10.24200/J40.2021.56291.1559

Sharif Mechanical Engineering Journal
Volume 37, Issue 1, Page 59-69, Original Article

- Received 26 August 2020; received in revised form 17 December 2020; accepted 8 January 2021.

Abstract

Magnesium alloys due to their compact hexagonal structure (HCP) usually have a yield asymmetry in tensile and compressive loadings. This is due to various active plastic deformation mechanisms under tensile and compressive loadings. Therefore, it is necessary to study the deformation behavior of magnesium alloys on the grain scale. This study focuses on the 3D Representative Volume Element (RVE) simulations of extruded magnesium alloy to investigate the effect of basal texture intensity on tensile-compressive yield asymmetry of Mg-0.8wt% Y. Polycrystalline aggregate models are built based on the theory of Voronoi diagram; then, the Slip-based crystal plasticity formulation is applied as a User Material Subroutine in ABAQUS (UMAT) to study the relationship of microstructures and stress-strain responses in Mg-0.8wt% Y alloy during tension and compression test and the results are presented.

The results show that when the orientation of the grains is considered random (zero texture intensity), the activities of different deformation modes are almost the same and minimum tensile-compressive yield asymmetry happens.

Also, the results reveal when all the grains are oriented in a certain direction (texture intensity 1), maximum tensile-compressive yield asymmetry occurs. Indeed, according to the Relative Activity (RA) results, in agreement with Schmid Factor (SF) results, in a strong basal texture with the intensity of 1, maximum contribution of accumulated plastic deformation is related to tensile twinning while in compressive loading, prismatic slip and contraction twinning are responsible for major plastic deformation. Therefore, due to the activation of various deformation modes in strong basal texture, the highest amount of tensile-compressive yield asymmetry occurs.

The S-shape strain hardening curve which is the main characteristic of twin-dominated deformation shows a strong dependency on texture in the extruded material so that during loading at the weaker texture, s-shape hardening is no longer present due to the dominated non-basal slips in weaker texture.

Key Words: Slip systems, tensile and contraction twinning, crystal plasticity, schmidt factor, basal texture.

NUMERICAL MODELING AND GEOMETRIC OPTIMIZATION OF SINGLE-PHASE SUPERSONIC EJECTOR

N. Nabatian(corresponding author)
n_nabatian@sbu.ac.ir
M. Shirazi
mehranshirazi237@gmail.com
S. Jafarzadeh Kenari
sjafarzadeh14@gmail.com
Dept. of Mechanical Engineering
Shahid Beheshti University
DOI:10.24200/J40.2021.57003.1567

Sharif Mechanical Engineering Journal
Volume 37, Issue 1, Page 71-78, Original Article

- Received 8 November 2020; received in revised form 27 January 2021; accepted 2 February 2021.

Abstract

In the present study, the performance of the 2D single-phase supersonic ejector with working fluid of air is simulated in ANSYS CFX. The aim is to investigate the velocity field, pressure distribution, primary nozzle flow regime, and entertainment ratio in different operational conditions. The primary pressure inlet with P=4 bar and the secondary inlet as an opening with P=1 bar at different outlet pressures are simulated. The $k-\varepsilon$ turbulence model is used. The sidewalls are considered symmetry boundary conditions and the no-slip condition is

M. Reihanian
mreyhanian@mech.sharif.edu
Dept. of Mechanical Engineering
Sharif University of Technology
DOI:10.24200/J40.2020.56029.1556

Sharif Mechanical Engineering Journal
Volume 37, Issue 1, Page 37-48, Original Article

- Received 11 July 2020; received in revised form 8 November 2020; accepted 15 November 2020.

Abstract

The main feature of RCCI is the significant reduction of nitrogen oxides and soot pollutants as efficiency and fuel consumption decrease. The aim of this study was to investigate the effects of the start of diesel fuel injection and injection pressure and two stage fuel injection strategies on RCCI combustion. The engine under investigation is a single cylinder engine with diesel fuel injection as high-reactivity fuel directly into the cylinder and natural gas fuel injection as low-reactivity fuel at the intake manifold. The start of injection, injection shape, and injection length in both injectors are controlled by the developed ECU. All tests were performed at an inlet air temperature of 37°C and the inlet pressure of 87kPa and 1800rpm. The results of this study show that by increasing diesel fuel injection pressure from 300 to 600 bar, at the 55° BTDC start of injection, the ignition starts 2.5 degrees earlier and the in-cylinder maximum pressure and HRR increase by 0.66% and 2.68%, respectively. By delaying the start of diesel fuel injection by 6 degrees, we see a 12.37 % reduction in maximum in-cylinder pressure, and the start of ignition is delayed to 5 degrees. Increasing the spray pressure of diesel fuel increases NOx by 154 % and decreases UHC and CO by 32 and 57%, respectively. Increasing the diesel injection pressure increases NOx by 154 % and decreases UHC and CO by 32 and 57 %, respectively. On the other hand, the results of the two-stage injection of diesel fuel show that by postponing the first start of injection, the in-cylinder maximum pressure, HRR, IMEP, and NOx emission increase and UHC and CO decrease. Also, by postponing the second start of injection, the in-cylinder maximum pressure, HRR, IMEP and NOx emissions are reduce,d while UHC and CO increase.

Key Words: Reactivity control compression ignition, injection pressure, multi-injection, thermal efficiency, pollution.

MULTI-OBJECTIVE OPTIMIZATION OF HYDRAULIC ENGINE MOUNTS VIBRATIONAL BEHAVIOR BY NON-DOMINATED SORTING GENETIC ALGORITHM

F. Fallahi
f.fallahi8@gmail.com
R. Shabani(corresponding author)
r.shabani@urmia.ac.ir
Gh. Rezazadeh
g.rezazadeh@urmia.ac.ir
S. Tariverdilo
s.tariverdilo@urmia.ac.ir
Faculty of Engineering
Urmia University
DOI:10.24200/J40.2021.56049.1557

Sharif Mechanical Engineering Journal
Volume 37, Issue 1, Page 49-58, Original Article

- Received 11 July 2020; received in revised form 24 January 2021; accepted 31 January 2021.

Abstract

Engine mounts are designed to hold the engine and isolate its vibration from the chassis of the vehicle. For optimum system performance, the mount must have high dynamic stiffness in the low-frequency range and low dynamic stiffness in the high-frequency range. As the conventional elastomeric mounts fail to satisfy such requirements due to their frequency-invariant behavior, the hydraulic engine mounts have been proposed, which provide appropriate dynamic stiffness employing two fluid-containing chambers. These two chambers are connected through a high-damped pass named inertia track and a floating plate named decoupler. In this paper, the low-frequency range response of a hydraulic engine mount has been studied using a discrete model. The effect of its parameters on the dynamic stiffness, damping ratio, and transmissibility of the mount is discussed. It is shown that increase in dynamic stiffness and damping ratio of the hydraulic engine mount is in contradiction with the decrease in its transmissibility. Hence, a multi-objective non-dominated sorting genetic algorithm has been used to achieve the desired results and the corresponding Pareto front is plotted. It is observed that upper chamber compliance and the effective area of the mount are the most dominant parameters in the optimization procedure. Also, a penalty function is used to transfer the maximum transmissibility out of the engine operation frequency range. Finally, the optimization results for transmissibility, damping ratio, and dynamic stiffness are presented for three series of parameters. It can be concluded that on the limit points of the Pareto front, which correspond to the one-objective optimization, the professed objective is optimized, but on the contradictory objective, no improvement is observed. There are points on the Pareto front where all the three objectives are optimized simultaneously. Therefore, as

temperature distribution is analyzed with plane strain assumptions. Then, the hoop stress distribution is determined. Finally, by assuming the small crack length and applying the superposition method, the problem of cracked viscoelastic cylinder is replaced by the problem of cracked viscoelastic cylinder subjected to determined average stress in the previous step. By employing the proper geometry factor, the stress intensity factor of the viscoelastic cylinder is determined. A parameter study is performed to investigate the effects of various parameters on the stress intensity factor of the cylinder. To validate the results, the problem of viscoelastic cracked cylinder is simplified to an elastic cracked cylinder by neglecting the viscoelastic terms, and the data obtained in the present work are compared with those given in the literature search. The results show that the proposed method is quite capable of estimating the stress intensity factor of small cracks in structures made of viscoelastic materials. A parameter study is performed to investigate effect of various parameters on stress intensity factor of the cylinder. The parameter study shows that a) the stress intensity factor of the viscoelastic cylinder in early moments of loading is higher than its equivalent elastic cylinder, b) increasing the elastic properties of the viscoelastic material results in higher stress intensity factor, and increasing the viscous properties of the material decreases the stress intensity factor.

Key Words: Stress intensity factor, viscoelastic cylinder, zener model, internal pressure , thermal distribution.

MINIMUM WEIGHT DESIGN OF TRUSS STRUCTURE VIA FORCE METHOD AND JAYA ALGORITHM

A. Barzegari
barzgariali@gmail.com
A. Alinia-Ziazi(corresponding author)
ali_alinia@srbiau.ac.ir
A. Maghsoudpour
a.maghsoudpour@srbiau.ac.ir
Dept. of Mechanical Engineering
Science and research branch, Islamic Azad University
DOI:10.24200/J40.2020.55805.1551

Sharif Mechanical Engineering Journal
Volume 37, Issue 1, Page 27-36, Original Article

- Received 8 June 2020; received in revised form 9 August 2020; accepted 23 August 2020.

Abstract

This research aims to minimize the weight of truss structures using force method formulation as a structural analyzer and Jaya algorithm as an optimizer tool. Constraints considered herein include stress limitations, displacement limitations, and size limitations. Design variables include the cross-sectional area of each element. They may easily be related to each other which will lead to decrease of design variables.

Jaya algorithm is a meta-heuristic random search method recently developed for constrained and unconstrained problems. The main superiority of Jaya algorithm to other random search methods is that it does not need any specific tuning parameter to generate next population. This algorithm consists of two steps in each cycle. First, a new population is generated using a simple random formula. Second, each new point is compared to its corresponding previous one while penalty function method is implemented. If the new point is in a better condition than the old one, the old one is replaced by the new one. All points are tested similarly till the population is updated. The procedure is repeated so as to achieve the desired convergence.

Several landmark examples appearing in the literature have been solved by the proposed method, thus showing the efficacy of the developed procedure. A perusal of results shows that procedure is not sensitive to the starting points and it should just be selected large enough to lie in feasible-usable design space. Moreover, rapid reduction of weight is obtained in the first few steps and the tendency of decreasing of the weight appears to be monotonic and uniform in all examples.

Unlike other metaheuristic methods that need a large number of optimization cycles to settle near the optimum point, combination of force method and Jaya algorithm provides higher computational efficiency and rapid convergence ability achieved by the above match. This is owing to the forced method formulation that makes the stress constraints to be linear, resulting in facilitating the procedure and enhancing its efficiency.

Key Words: Force method, jaya algorithm, metaheuristic, truss structures, random search.

EXPERIMENTAL INVESTIGATION OF THE EFFECT OF PILOT FUEL INJECTION PRESSURE AND START OF INJECTION ON COMBUSTION AND EMISSION OF RCCI ENGINE FUELED WITH DIESEL-CNG

S. Ghaffarzadeh
ghaffarzade@gmail.com
A. Nassiri Toosi(corresponding author)
anasiri@ardakan.ac.ir
Dept. of Automotive Engineering
Iran University of Science and Technology

Abstracts of Papers in English

NUMERICAL INVESTIGATION OF TRANSIENT TEMPERATURE SEPARATION PHENOMENON IN VORTEX TUBE

M. Mirjalili
mr_mirjalili@yahoo.com
K. Ghorbanian(corresponding author)
ghorbanian@sharif.edu
Dept. of Aerospace Engineering
Sharif University of Technology
DOI:10.24200/J40.2020.54808.1547

Sharif Mechanical Engineering Journal
Volume 37, Issue 1, Page 3-15, Original Article

- Received 4 April 2020; received in revised form 1 November 2020; accepted 15 November 2020.

Abstract

Ranque-Hilsch vortex tube is a simple mechanical device with no moving parts. A high pressure feed gas enters the vortex tube through the swirl nozzles causing the feed gas to split into two thermodynamically different streams. These two streams will be having not only different velocities but also distinguished temperatures that are lower and higher than the inlet feed gas temperature. This phenomenon and the associated energy separation of the feed gas through the vortex tube are strongly dependent on such parameters as geometry, position, and number of the swirl nozzles, diameter and length of the vortex tube, inlet feed gas pressure, control valves, and aperture duct size. Although the vortex tube is used for few decades across different industries, energy separation phenomenon is still neither fully explained nor agreed upon by the scientific community. This paper is an attempt at a better physical understanding of the embedded phenomenon using computational fluid dynamics via a commercial software (Fluent Software) to numerically simulate the transient flow behavior of the feed gas as well as the energy separation, resulting in distinguished gas streams in a two-dimensional and axisymmetric vortex tube. Appropriate boundary conditions are employed in the numerical simulation resembling experimental conditions from the open literature with the exception of the gas exit at the hot end which has been set to be in concert with the operation of the flow-control valve. The obtained numerical results are in good agreement with experimental data from the open literature. Further, the numerical simulation confirms the existence of free and forced vortices and indicates that the temperature of the circumferential elements towards the hot end gets hotter by receiving heat from the core flow due to the kinetic to thermal energy conversion in the presence of viscous shear stresses.

Key Words: Vortex tube, transient solution, temperature separation, cold mass fraction, computational fluid dynamics.

STRESS INTENSITY FACTOR FOR A SMALL RADIAL CRACK LOCATED ON EXTERNAL EDGE OF A ROTATING VISCOELASTIC CYLINDER SUBJECTED TO INTERNAL PRESSURE AND TEMPERATURE DISTRIBUTION

R. Nikpour
rasoulnikpour@yahoo.com
H. Mahbadi(corresponding author)
h_mahbadi@iauctb.ac.ir
Dept. of Mechanical Engineering
Central Tehran Branch, Islamic Azad University
DOI:10.24200/J40.2020.54689.1539

Sharif Mechanical Engineering Journal
Volume 37, Issue 1, Page 17-26, Original Article

- Received 25 April 2020; received in revised form 28 September 2020; accepted 8 November 2020.

Abstract

In this research, stress intensity factor of a small radial crack in a rotating thick cylinder made of viscoelastic materials subjected to internal pressure and radial temperature distribution is investigated. The radial crack is assumed to be located at the outer edge of the cylinder. The Zener model (i.e., standard three parameter solid) is applied to simulate the viscoelastic behavior of the cylinder. To obtain the stress intensity factor of the viscoelastic cylinder, the problem of uncracked viscoelastic cylinder subjected to internal pressure and radial