Mathematical Feature of Elliott Wave in Foreign Currency Trend

Naoyasu Kita
Faculty of Advanced Science and Technology, Kumamoto University,

Corresponding author: Naoyasu Kita
E-mail: nkita@kumamoto-u.ac.jp

Keywords: Rate of Currency, Elliott Wave Principle, Optimization, Method of Penalty Functions

Abstract

Today, the FX (initial letters of "Foreign eXchange") is of interest among many finance traders who want to produce profits by using trends of exchange rate of foreign currencies (see Fig.1). "If there are laws or principles in the fluctuations, the forecast of the trend will become easier." Some groups of FX traders had been wondering, and they noticed that the **Elliott wave** process sometimes took place in the fluctuation. The wave is named after Ralph Nelson Elliott (1871–1948). He proposed that **market prices unfold in specific patterns like a fractal**, which practitioners today call "Elliott waves". Some FX traders have applied this principle to the forecast of the trend. The Elliot wave consists of fractal-like pattern (see Fig.2). Therefore, once a small basic pattern (called the **Elliot sub-wave**) is detected, a larger pattern will possibly emerge next. Then it is important to detect a sub-wave more swiftly than anyone else.

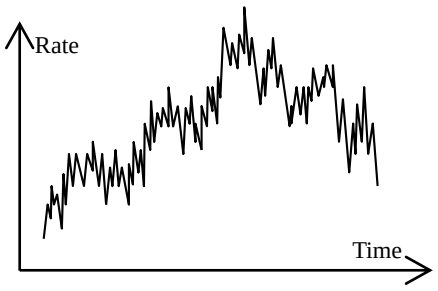


Fig. 1 Trend of Exchange Rate

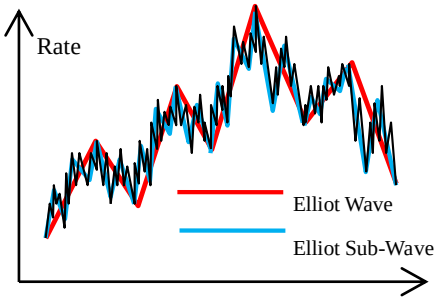


Fig. 2 Approximation by Line Graph

In this talk, I will introduce an idea of automated detection of Elliott sub-wave. The key is to construct a well-approximated line graph. Since the Elliot wave holds special constraints, the mathematical tool called "**nonlinear optimization**" is applicable to the detection of **Elliot sub-waves**. In addition, if time permits, I will introduce how to make sure whether the resulting line graph is appropriate or not.

Naoyasu Kita *received the B.S. degree in science from Waseda University, Japan, in 1993, M.S. degree in science from Tokyo University, Japan, in 1995 and Ph. D. degree in mathematical science from Nagoya University, Japan, in 1998. He is currently a professor in Faculty of Advanced Science and Technology, Kumamoto University. His research interests include pure mathematics – in particular, Theory of Partial Differential Equation, Functional Analysis, Fourier Analysis and Real Analysis.*

On the uniqueness of adapted solutions to BSDEs

Yufeng Shi
School of Mathematics and Institute for Financial Studies, Shandong University

Corresponding author: Yufeng Shi
E-mail: yfshi@sdu.edu.cn

Keywords: Multidimensional BSDE, adapted solution, envelope, Girsanov's theorem

Abstract

In the theory of backward stochastic differential equations (BSDEs in short), in order to guarantee the existence and uniqueness of adapted solutions to BSDEs, one usually supposes that the generator g is Lipschitz with respect to y and z or other similar hypothetical conditions in which the variable z satisfies the similar assumptions as the variable y. In this talk, we show that BSDEs also has a unique adapted solution if the generator g is Lipchitz with respect to y and is linear growth and continuous with respect to z. This implies that the variable z is determined by y. And as an application, we obtain that the corresponding partial differential equations have a unique viscosity solution.

Yufeng Shi received the B. S., M. S. and Ph.D degrees in Mathematics from Shandong Normal University, China, in 1992, and from Shandong University, China, in 1995 and 1998, respectively. He is currently a professor in the School of Mathematics and Institute for Financial Studies, Shandong University. His research interests include Backward Stochastic Differential Equations, Stochastic Partial Differential Equations, Nonlinear Mathematical Expectations, Stochastic Analysis, Stochastic Control, Probability and Statistics, Mathematical Finance and Quantitative investment.

On the relationship between combinatorial structures of subset sum problems and generalized dimensions

Shoichi Kamada
Department of Applied Mathematics,
Graduate School of Science and Technology, Kumamoto University

Corresponding author: Shoichi Kamada
E-mail: 168d9303@st.kumamoto-u.ac.jp

Keywords: Fractal Analysis, Multi-fractal Analysis, Subset Sum Problems, Linear Diophantine Equations, Combinatorics, Cryptography

Abstract The subset sum problem is a combinatorial problem in mathematics. Since the subset sum problem is NP-hard, it has several applications to cryptography. It is well-known that a cryptosystem based on the subset sum problem, for example, a knapsack cryptosystem, is easy to attack when the density of the problem is low.

In this talk, we introduce a framework in order to deal with fractal and multi-fractal analysis for subset sum problems. To do this, we define non-classical generalized dimensions for a family of subset sum functions. Each of such generalized dimensions requires a q-fractal dimension, which can be defined for an arbitrary subset sum function in the family, and some embedding into the 1-dimensional Euclidean space plays an important role. This enables us to investigate (multi-)collisions for a subset sum function from some combinatorial structure via fractal and multi-fractal analysis. Moreover, these non-classical generalized dimensions not only include the box-counting dimension, the information dimension and the correlation dimension, but also include the density of the subset sum problem. We give a lower bound for a q-fractal dimension at q=0 (box-counting).

This talk is a part of arXiv:1808.03033.

Shoichi Kamada received the Bachelor of Engineering and the Master of Engineering degrees from Kumamoto University, Japan, in 2014 and 2016, respectively. He is currently a doctoral student in the Department of Applied Mathematics, Kumamoto University. His research interests include number theory, combinatorics, fractal analysis and cryptography.

Maximal Entropy Random Walk on heterogenous network for MiRNA-disease Association prediction

Ya-Wei Niu and Guang-Hui Wang
School of mathematics, Shandong University, Jinan 250100, China
Corresponding author: Guang-hui Wang
E-mail: ghwang@sdu.edu.cn

Keywords: maximal entropy random walk; heterogenous network; microRNA; disease; miRNA-disease association

Abstract

The last few decades have verified the vital roles of microRNAs in the development of human diseases and witnessed the increasing interest in the prediction of potential disease-miRNA associations. Owning to the open access of many miRNA-related databases, up until recently, kinds of feasible in silico models have been proposed. In this work, we developed a computational model of Maximal Entropy Random Walk on heterogenous network for MiRNA-disease Association prediction (MERWMDA). MERWMDA integrated known disease-miRNA association, pair-wise functional relation of miRNAs and pair-wise semantic relation of diseases into a heterogenous network comprised of disease and miRNA nodes full of information. As a kind of widely-applied biased walk process with more randomness, MERW was then implemented on the heterogenous network to reveal potential disease-miRNA associations. Cross validation was further performed to evaluate the performance of MERWMDA. As a result, MERWMDA obtained AUCs of 0.8966 and 0.8491 respectively in the aspect of global and local leave-one-out cross validation. What' more, three different case study strategies on four human complex diseases were conducted to comprehensively assess the quality of the model. Specifically, one kind of case study on Esophageal cancer and Prostate cancer were conducted based on HMDD v2.0 database. 94% and 88% out of the top 50 ranked miRNAs were confirmed by recent literature, respectively. To simulate new disease without known related miRNAs, Lung cancer (confirmed ratio 94%) associated miRNAs were removed for case study. Lymphoma (verified ratio 88%) was adopted to assess the prediction robustness of MERWMDA based on HMDD v1.0 database. We anticipated that MERWMDA could offer valuable candidates for in vitro biomedical experiments in future.

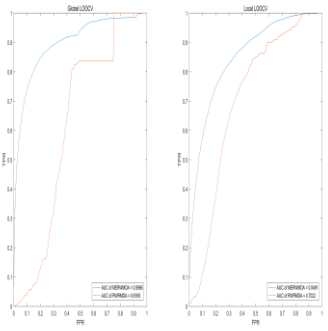


Figure1 Performance comparison between MERWMDA and RWRMDA in global and local LOOCV

Author 1 Yawei Niu received the B.S. degree in Information and Computing Science from School of Mathematical Sciences, Dalian University of Technology in 2014. From September 2015 to now, he is a Master Student in School of Mathematics, Shandong University. His current research interests include graph theory, complex networks, bioinformatics and their application.

Author 2 Guanghui Wang received a B.S. degree in Mathematics from Shandong University, China, in 2001. He got the doctor's degree from Paris sud University and worked in Ecole Centrale Paris as a Post-doctor. Now he is a professor in School of Mathematics, Shandong University. His current interests include graph theory, combinatorics, complex networks and bioinformatics.

Photo (3 cm ×2.5 cm)

Photo (3 cm ×2.5 cm)

Degree sum conditions for partitioning graphs into cycles with a specified number of chords

Shuya Chiba[1], Suyun Jiang[2] and Jin Yan[3]
[1]*Faculty of Advanced Science and Technology, Kumamoto University*
[2]*Institute for Interdisciplinary Research, Jianghan University*
[3]*School of Mathematics, Shandong University*

Corresponding author: Shuya Chiba
E-mail: schiba@ kumamoto-u.ac.jp

Keywords: Partitions, Cycles, Chorded cycles, Degree sum

Abstract

We consider finite simple graphs, which have neither loops nor multiple edges. In the following, "partition" and "disjoint" always mean "vertex-partition" and "vertex-disjoint", respectively.

The classical Ore's theorem (Amer. Math. Monthly 1960) states that every graph of order $n \geq 3$ with degree sum at least n for any pair of non-adjacent vertices, is hamiltonian. Later, Brandt, Chen, Faudree, Gould and Lesniak (J. Graph Theory 1997) generalized the result by showing that the Ore condition (the degree sum condition) guarantees the existence of a partition of a graph into a prescribed number of disjoint cycles for sufficiently large graphs. In this study, we give a further generalization of these results by considering a partition into a prescribed number of disjoint subgraphs, each of which is a relaxed structure of a complete graph.

For a positive integer c, a cycle in a graph is called a *c-chorded cycle* if there are at least c edges between the vertices on the cycle that are not edges of the cycle. Since a Hamilton cycle of a complete graph of order t has exactly $t(t-3)/2$ chords, we can regard a c-chorded cycle as a relaxed structure of a complete graph of order t for $c = t(t-3)/2$. Concerning the existence of a partition into such structures, we show that the Ore condition also guarantees the existence of a partition of a graph into a prescribed number of disjoint c-chorded cycles for sufficiently large graphs.

This work was supported by JSPS KAKENHI Grant Number 17K05347 and by Grant for Basic Research Projects from the Sumitomo Foundation (grant number 170149).

Shuya Chiba received the B.S., M.S. and D.S. degrees in mathematics from Tokyo University of Science, Japan, in 2005, 2007 and 2010, respectively. He is currently an Associate Professor in the Faculty of Advanced Science and Technology, Kumamoto University. His research interests include packing and partitioning problem in graphs.

An Intelligent Mold Testing Method Developed for Injection Molding

Ming-Shyan Huang, Yung-Chih Fang
Department of Mechanical and Automation Engineering, National Kaohsiung University of Science and Technology, TAIWAN
Shih-Chih Nian
Department of Power Mechanical Engineering, National Taitung College, TAIWAN

Corresponding author: Ming-Shyan Huang
E-mail: mshuang@nkust,edu,tw

Keywords: Injection molding, polymer processing, process monitoring, scientific molding

Abstract

Traditional injection mold testing mainly relies on engineers' experiences to adjust the operation parameters of injection molding machines, focusing on the machine feasibility of molding quality parts. However, the obtained machine parameters are incapable of well controlling part quality when polymer melt is varying in mass-production. This study thereby proposes a scientific molding approach by installing various sensors in the cavity to detect the condition of polymer melt in the process of mold filling. The main idea is to control injection-molding quality by referencing to melt polymer condition instead of machine condition. By applying domain know-how on analyzing the signal profiles from these sensors, this study has established a standard procedure and rules for molding testing. In addition, this study has developed a robust parameter profile by computational learning inspected qualities. Such a method enhances a high yield rate in mass production and is superior to conventional mold testing. The proposed intelligent mold testing comprises of determining V/P switchover time, gate-frozen time, and stages of holding that generates minimal shrinkage deviation. By employing IC tray as a research vehicle, the warpage of injection-molded parts are examined to verify the feasibility of the intelligent mold testing. Experimental results show that the warpage of IC tray is successfully reduced from 0.62 mm to 0.40 mm, whereas the sintered quality is also consistently improved.

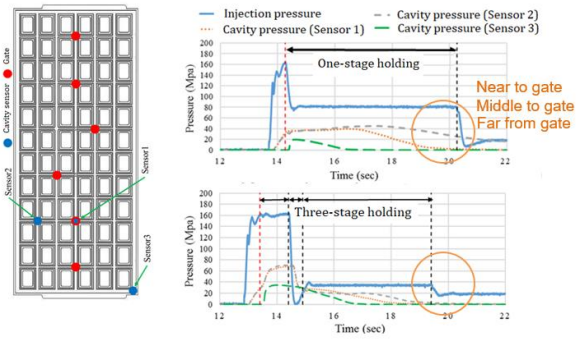


Fig. 1. Stages of holding pressure and time

Ming-Shyan Huang received the B.S. in Mechanical Engineering from National Taiwan University, Taiwan, in 1987, the M.S. in System Engineering from Case Western Reserve University, USA, in 1991, and the Ph.D. degree in Mechanical Engineering from University of Wisconsin-Madison, USA, 1994. He is currently a full professor in the Department of Mechanical and Automation Engineering, National Kaohsiung University of Science and Engineering. His research interests include injection molding, system dynamics and control, and industrial design.

Machine learning-based context-aware application in manufacturing

Yingxin Ye and Tianliang Hu
School of Mechanical Engineering, Shandong University

Corresponding author: Tianliang Hu
E-mail: tlhu@sdu.edu.cn

Keywords: context-aware, machine learning, manufacturing

Abstract

Modern factory applies automation equipment with fewer workers. However, with more equipment and the data they collected, it will be difficult for workers to react in time with the information they are interested.

A context-aware system can use context to provide relevant information and/or services to the user, which can solve aforementioned problem.

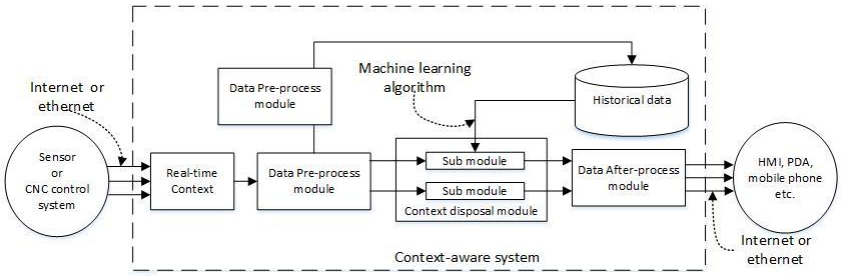


Fig 1. Context-aware system structure

This study assumes a context-aware application scenario in manufacturing and designs the machine-learning-based context-aware system as shown in Fig1. 5 different machine learning algorithms are applied into simulated manufacturing data to test their performance both on correct rate and false positive rate. Feasibility of applying machine learning into context-aware manufacturing system has been proved and a general development method is introduced.

Yingxin Ye received the B. E. degrees from Shandong University, China, in 2013. She is currently a doctoral student in the School of Mechanical Engineering, Shandong University. Her research interests include intelligent manufacturing, CNC technology

Tianliang Hu received the B. E. and Ph. D. degrees in Mechanical Engineering from Shandong University, China, in 2003 and 2008, respectively. He is currently an associate professor in the School of Mechanical Engineering, Shandong University. His research interests include intelligent manufacturing, Cyber-Physical system, CNC technology, and robotic technology

DVC measurement of invasive deformation field of ECM generated by cancer cell with/without EMT

Yasuyuki Morita
Faculty of Advanced Science and Technology, Kumamoto Univerity

Corresponding author: Yasuyuki Morita
E-mail: ymorita@kumamoto-u.ac.jp

Keywords: Cancer, Metastasis, Epithelial-mesenchymal transition, Extracellular matrix, Digital volume correlation

Abstract

Eradication of cancer is one of the most important dreams which people all over the world wish to resolve as soon as possible. Metastasis is a vital key to death from cancer. That is, it is inevitable to understand the mechanism of cancer metastasis and then to control/suppress it. Cancer metastasis is a type of mechanical interactions between cancer cells and extracellular matrix (ECM) which cancer cells go through ECM by pushing and pulling the fibers of ECM. Therefore, elucidating the mechanical interactions is the first step to get the picture of cancer metastasis. Here, I measured a deformation field of ECM exerted by cancer cell, which I call it "invasion dynamics field," using a digital volume correlation (DVC) method. The change of the invasion dynamics field before/after epithelial-mesenchymal transition (EMT) was evaluated. As a result, traction force of an invasive cancer cell (Hela) was dramatically increased by EMT (Fig. 1), while invasive dynamic field of a non-invasive cancer cell (MCF-7) was almost same before/after EMT. This result suggests that the mechanical interaction between cancer cell and EMC was promoted by EMT in invasive cancer cells, and the cells get higher ability to metastasize.

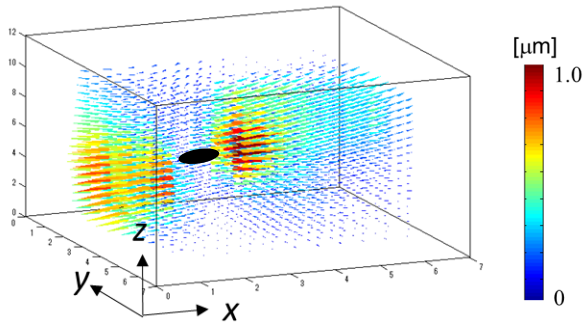


Fig. 1 One representative example of the invasion dynamics field of a cancer cell. The figure shows the displacement vector field of type I collagen gel surrounding a Hela cell. The black ellipsoid indicates a Hela cell whose size is approximately 80 μm.

Yasuyuki Morita received the Ph.D. in materials science from Kyushu University, Japan, in 2003. He is currently a professor in the Faculty of Advanced Science and Technology, Kumamoto University. His research interests include biomechanics and experimental mechanics.

Introduction of Multi-Fluids Mixer and Its Application to Water Purification as Microbubble Generator

Akimaro Kawahara
Faculty of Advanced Science and Technology, Kumamoto University

Corresponding author: Akimaro Kawahara
E-mail: akimaro@mech.kumamoto-u.ac.jp

Keywords: Multi-fluids mixer, Microbubble, Mists, Water purification

Abstract

The author invented a multi-fluids mixer with multifunction, which can generate microbubble, mist (i.e., tiny liquid droplet), and emulsion of immiscible liquids. In this presentation, the outline and the hydraulic performance of the mixer studied to date in Kumamoto University are described especially for its usage as the microbubble generator, with its application to water purification. In the treatments of waste water, sewage water and contaminated water etc., biochemical treatments are often utilized. However, the improvement of treatment performance is required with the lower cost and easy maintenance. To promote the biochemical reaction by aerobic bacterium, the oxygen supply rate into microorganism has to be fast. Thus, effective oxygen supply method is essential for the aerobic water treatment. Microbubbles are tiny bubbles, less than a few hundred micrometers in diameter, and have several characteristics, such as high dissolubility in water around them. Therefore, the microbubbles have an advantage to dissolve the oxygen gas in air into water. In this connection, the effects of microbubbles on water purification performance were studied. To confirm the superiority of micro-bubble as an aeration system, a comparative test of microbubble generator and an air sparger was conducted. Volumetric mass transfer coefficients were measured for respective aeration methods. As a water purification test, an artificial sewage water was aerated with the micro-bubble generator and the air sparger during three weeks. A result of such tests and a practical significance of the purification system using the micro-bubble are reported in this presentation.

Fig. 1 Micro-bubbles generated by multi-fluids mixer.

Akimaro Kawahara is a professor of mechanical engineering at Kumamoto University, Japan. He received his doctoral degree in 1998 from Kumamoto University. His research is concerned with multiphase flow, especially gas–liquid two-phase flow. So, his current research topics cover the improvement of subchannel analysis code for predicting thermo-hydraulic behavior of coolant in BWR, the industrial applications of a multi-fluids mixer invented by him which can generate micro-bubbles, mist and emulsion, etc., and the derivation of the correlations of two-phase flow parameters applicable to micro- and mini-channels.

Development of a Novel Heat-transfer Device Using a 3-D Printer

Yasushi Koito[1] and Fumihiko Hideyama[2]
[1] *Division of Industrial Fundamentals, Kumamoto University*
[2] *Department of Advanced Mechanical Systems, Kumamoto University*

Corresponding author: Yasushi Koito
E-mail: koito@gpo.kumamoto-u.ac.jp

Keywords: Thermal Engineering, Phase-change Heat Transfer, Heat Pipe, 3-D Printer

Abstract

3-D printing technology has been widely utilized for fabrication of 3-D objects. The 3-D printer enables the fabrication of objects directly from digital data of CAD. Besides, new design of objects can be achieved without difficulty.

In our group, the attempt has been made to develop a novel heat pipe using the 3-D printer. The heat pipe is a thermal device that transports heat from one position to another with no additional power input. The heat pipe has been used in many ways, especially for cooling of electronic devices. Fig. 1 shows the heat pipe channel fabricated by the 3-D printer. Fig. 2 shows the 3-D printed heat pipe walls before and after surface treatment. Experiments are also conducted to evaluate the heat transfer characteristics of the 3-D printed heat pipes.

Fig. 1 3-D printed heat pipe channel

Fig. 2 Heat pipe walls before (left) and after (right) surface treatment

---

Yasushi Koito received the M. E. and D. E. degrees from Kyushu University, Japan in 1999 and 2002, respectively. He is currently an associate professor at Division of Industrial Fundamentals, Faculty of Advanced Science and Technology, Kumamoto University. His research interests include a heat pipe, a vapor chamber, a 3-D printer and a numerical simulation.

Fumihiko Hideyama received the M. E. degree from Kumamoto University, Japan in 2016. He is currently a doctoral course student at Department of Advanced Mechanical Systems, Graduate School of Science and Technology, Kumamoto University. His research interests include a vapor chamber and a 3-D printed pulsating heat pipe.

Proceedings of the Engineering Workshop 2018
November 15-17, 2018, Kumamoto University, Japan

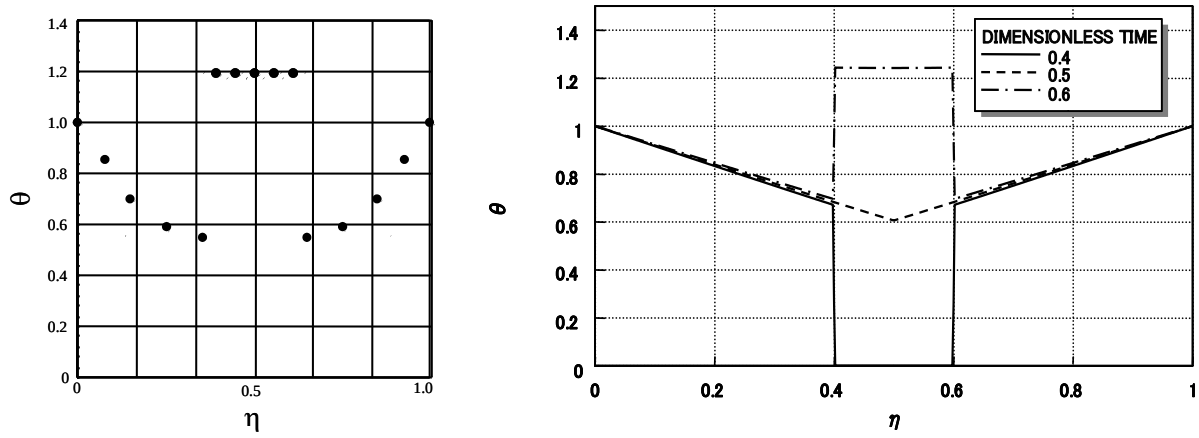
Thermal Propagation in Thin Film Heated from Both Side-Wall

Shuichi Torii
Department of Mechanical and Mathematical Engineering, Kumamoto University
E-mail: torii@mech.kumamoto-u.ac.jp

Keywords: thermal propagation, thin film, Hyperbolic Equation, numerical simulation

Abstract

The present study is performed on thermal propagation in a very thin film subjected to a symmetrical temperature change on both sides by means of molecular dynamics method. Numerical result is compared with the other numerical one which is obtained from the non-Fourier, hyperbolic heat conduction equation using a numerical technique based on MacCormak's predictor-corrector scheme. Consideration is given to the time history of thermal wave before and after symmetrical collision of wave fronts from both sides of a film. It is disclosed that (i) in transient heat conduction, thermal wave front is transported as a wave in the film, (ii) substantial temperature amplification causes within a very short period of time, and (iii) the heat-affected and undisturbed zones are found to be caused by the atomic movement and in particular the temperature overshoot is induced by the substantial amplification of the atom movement in the film (Figure1.).



(a) (b)

Figure 1 Temperature distribution after collision of thermal wave: (a) molecular dynamics method and (b) non-Fourier, hyperbolic heat conduction equation.

Shuichi Torii received the B.D. degree from Kagoshima University in 1983 and M.D. and Ph.D. degree from Kyushu University in 1985 and 1989, respectively, all Mechanical Engineering. He then worked as the visiting scholar at University of Michigan. Since 2003, he currently is a Professor of Department of Mechanical Engineering at Kumamoto University. He focuses on production and development of clean Energy and renewable Energy, thermal fluid flow transport phenomena using nanofluids, and development of new clean fuel with the aid of shock-wave.

Study on dominant parameters of probe design for wafer test

Shunta Mori*, Keisuke Takeda*, Yasuo Marumo**
*Graduate School of Science and Technology, Kumamoto University, Japan
**Faculty of Advanced Science and Technology, Kumamoto University, Japan

Corresponding author: Shunta Mori
E-mail: 173d8562@st.kumamoto-u.ac.jp

Keywords: Wafer test, Contact probe, Contact conditions, Friction

Abstract

In wafer tests, testing probes are indented into Si wafer. Smaller damage as well as stable electrical conduction are required for subsequent processes. In addition to appropriate test conditions, optimization of probe shape is a main factor for achieving them. Many testing probes are embedded in a probe card and its structure has various types. Furthermore, the testing probes have various sizes and geometries. Such probe cards and testing probes should be highly precise. However, so far their size and geometry have been designed based on workers' experiences. The design criteria and methods have been demanded for years. In this study, several prospective dominant parameters for probe design were investigated. Surface expansion, bulge height, indentation load, contact area and apparent coefficient of friction were confirmed to be effective parameters. The method used here can be applied to design for various types of probe cards and testing probes.

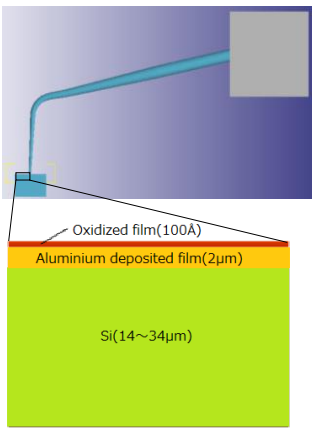


Fig. 1 Schematic illustration of probe indentation

Shunta Mori received the B. E. degrees in engineering from Kumamoto University, Japan, in 2016. He is currently a student in the Graduate School of Science and Technology, Kumamoto University.

Random Finite Element Analysis on Solid Rocked Engine Vessel

Chong LIU and Shuichi TORII
Department of Advanced Mechanical Systems, Kumamoto University
Kumamoto University, Kurokami, 2-39-1, Kumamoto 860-8555, Japan

Corresponding author: Shuichi TORII
E-mail: torii@mech.kumamoto-u.ac.jp

Keywords: solid rocket engine, shell, ANSYS, reliability

Abstract

In order to expand the scope of attack, both increase specific impulse and decrease weight are become the research goal. In this paper, we take the vessel of solid rocket engine as research object, simulate the pressure of hydrostatic test, establish the finite element model of solid rocket engine vessel, solve the stress for any element in solid rocket engine vessel and find the maximum stress point by ANASYS, and then we solve reliability by the method of random analysis. According to this reliability, change the pressure which load on the inside of solid rocket engine vessel while insure reliability higher than the criticality. In this analysis, we can increase the pressure which loads the solid rocket engine vessel and we achieve the sensitivity of all parameters.

In the present study, the solid rocket engine vessel is reliable or safe under 17MPa, but is conservative. Under the acceptable reliability, we can increase the pressure and reach 18.5 MPa. We can find which parameter is serious influence reliability of solid rocket engine vessel or not. It is theoretical base for optimal structure of solid rocket engine vessel. By comparing the method of random analysis with conventional method, we can find the method of random analysis is significant for designing solid rocket engine vessel.

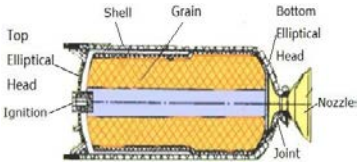


Fig. 1 Structure of solid rocket engine

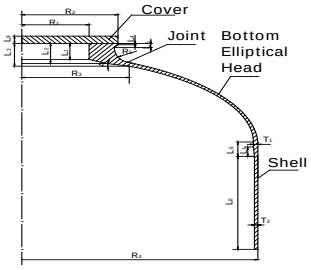


Fig.2 Simply structure of solid rocket engine vessel

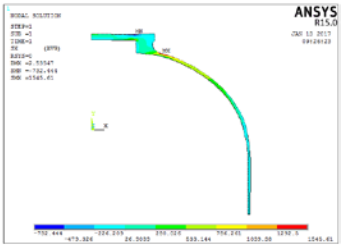


Fig.3 Stress contour plot under 18.5MPa

Chong LIU received the B. E. degree in Mechanical Design, Manufacturing and Automation from Liaoning Shihua University, China, in 2006 and the M. E. degree in Power Engineering and Engineering Thermophysics from Inner Mongolia University of Technology, China, in 2009. He is currently a doctoral student in the Department of Advanced Mechanical Systems, Kumamoto University. He research on the combustion characteristics of unused biomass

Induction coil modular design and mold surface heating application

Chun-Ying Liu and Ming-Shyan Huang
Department of Mechanical and Automation Engineering, Kaohsiung University of Technology

Corresponding author: Chun-Ying Liu
E-mail: u0614802@nkust.edu.tw

Keywords: Injection molding, induction heating, magnetic shielding technology, modular coil

Abstract

Higher mold temperature not only facilitates flow behavior during filling stage but also enhances surface quality of injection molded part. However, it leads to longer cooling time. Varitherm technology is demanded to improve productivity. Advantages of excellent heat efficiency and energy saving are possessed by induction heating. However, defects such as lower heat efficiency and non-uniform distribution of mold temperature is due to interference and rejection between magnetic fields caused by poorly designed coils. A large heating curved surface is investigated in this study. Modularized heating coils is designed to reduce heating area and required electric power to prevent proximity effect. In addition, magnetic field insulation and shielding method are also investigated in this research. Moreover, heating effect and the improvement in proximity effect by magnetic shielding through magnetic conductors are analyzed by computer aided of engineering, COMSOL. In simulation results, the energy conversion efficiency is reached to 2.59 by the integration of optimized coil design and magnetic shielding method. Besides, in experimental results the heating effect on regional area could be bettered by modularized heating coil designs since the mold temperature at central area is improved from 61.0 to 106.2℃. By this way, regional mold temperature is also easily controlled.

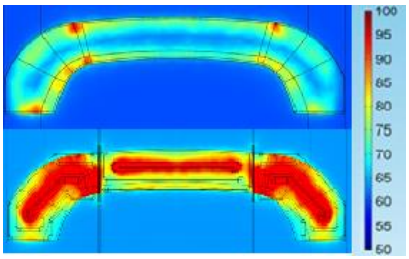


Fig. 1 COMSOL simulates heating effect on mold surface

This study intends to induction heat a large area of the curved surface, but the proximity effect will increase as the induction heating area increase. Therefore, the induction coil is modularized to reduce the heating area of each area and the small electric power induction heating machine is used is the focus of this study, and further research an inductive coil designed, insulated method and magnetic shielding technology of induction heating technology. In particular, the matching relationship between the induction coil and the heating machine is discussed. In this study, the electromagnetic coupling analysis software of COMSOL is used to simulate the matching relationship between the coil shape and the heating machine, and the modular coil is designed by using the analysis technology. The multiple coils are combined to achieve the same heating result as the large induction heater to reduce the cost of induction heating. The simulation results show that the optimized induction coil combined with magnetic shielding technology can achieve a relative energy conversion efficiency of 2.59. The results of high-frequency induction heating show that after using the modular coil, the central low temperature region of the mold rises from 61.0 to 106.2, modularized. The coil can effectively enhance the induction

heating effect of the area, make the temperature of the mold surface easier to control, and have the characteristics of heating a specific area, so that the surface temperature is not limited by the shape of the coil, and cannot be heated to certain specific areas.

Chun-Ying Liu received the B. E., degrees in Mechanical and Automation Engineering from Kaohsiung University of Technology, Taiwan, in 2017. He is studying for master's degree in the Department of Mechanical and Automation Engineering, Kaohsiung University of Technology.

Ming-Shyan Huang received the B.S. in Mechanical Engineering from National Taiwan University, Taiwan, in 1987, the M.S. in System Engineering from Case Western Reserve University, USA, in 1991, and the Ph.D. degree in Mechanical Engineering from University of Wisconsin-Madison, USA, 1994. He is currently a full professor in the Department of Mechanical and Automation Engineering, National Kaohsiung University of Science and Engineering. His research interests include injection molding,

Modal couple effect on milling stability of thin-walled workpiece

Jian Gao[1, 2], Qinghua Song[1, 2, 3] and Zhanqiang Liu[1, 2, 3]
[1]*School of Mechanical Engineering, Shandong University, Jinan 250061, P R China*
[2]*Key Laboratory of High Efficiency and Clean Mechanical Manufacture, Ministry of Education, China*
[3]*National Demonstration Center for Experimental Mechanical Engineering Education, Shandong*

Corresponding author: Qinghua Song
E-mail: ssinghua@sdu.edu.cn

Keywords: Modal couple effect, thin-walled workpiece, stability prediction, milling

Abstract

Milling chatter is one of the biggest obstacles to thin-wall milling, and accurate stability prediction method is an effective way to avoid milling chatter. The dynamic system of thin-walled workpiece milling has the characteristics of location-dependent and modal aggregation, which means that multiple modes are close with each other. The first few modes may be simultaneously excited by the cutting forces during milling to jointly dominate the milling response of the thin-walled plates. In this case, the existing single mode method (SMM) can't predict milling stability accurately. Additionally, multiple modes couple effect plays an significant impact on thin-walled component milling process in some cases. To reveal these problems, a modal coupled method (MCM) is proposed to consider this effect in the paper. Meanwhile, the validity and accuracy of MCM in the prediction of milling stability of thin-walled parts are verified by comparing MCM with the existing multi-mode method- lowest envelope method (LEM). Finally, thin-walled plate milling tests are performed to verify the agreement between the numerical and experimental results.

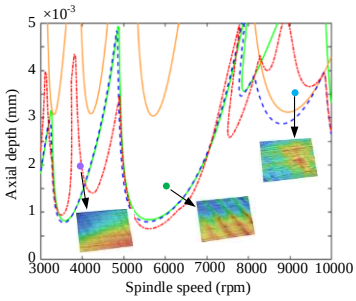


Fig. 1 SLD of system based on SMM, LEM and MCM

Jian Gao received the B.S. degrees in mechanical engineering from Shandong University, China, in 2016. Since 2016, he has been a M.S. student at Shandong University, China. His research interests include the plate vibration control and vibration signal analysis.

Qinghua Song received the B.S., M.S. and Ph.D. degrees in mechanical engineering from Shandong University, China, in 1999, 2003 and 2009. He is currently an associate professor of school of mechanical engineering, Shandong University. His research interests include structural vibration; (micro) cutting system dynamics; micro process equipment technology; biomedical machinery.

Zhanqiang Liu received the B.S. and M.S. degrees in mechanical engineering from Shandong University, China, in 1994 and the Ph.D. degree in philosophy from City University of Hong Kong, Hong Kong, China, in 1999. He is currently a professor of school of mechanical engineering, Shandong University. His research interests include machining theory and tool technology.

Utilization of CFD Simulation to Realize the Mixing Behavior of Liquid Sludge in An Anaerobic Digester

Huqe Farhana and Shuichi TORII
Department of Advanced Mechanical System, Kumamoto University

Corresponding author: Huqe Farhana
E-mail: farhanahuqe@gmail.com

Computational Fluid Dynamics (CFD), Turbulence model, Scalar field pattern, Relative total temperature, Turbulent viscosity

Abstract

The importance of utilization of biogas as an alternative of fossil fuel is not a new idea. But still now the conditions of producing maximum biogas are not optimized perfectly. Because the system is opaque and it is very tough to observe the digestion through open eyes. In such case, computational fluid dynamics (CFD) simulation has been used as a powerful technique to understand the digestion process in terms of scalar field and vector field pattern of some selective parameters, which are very important to understand the mixing behavior of the raw materials and thus the simulated output is very helpful to optimize the conditions for biogas production by anaerobic digestion. The main reason of such situation is the lacking of understanding mixing behavior of the raw materials inside the anaerobic digester. Though Anaerobic Digestion Model1 (ADM1) is one of the convenient models to realize the behavior of waste materials, but the 3D flow visualization of computational fluid dynamics (CFD) can make the comprehension more specific and realistic. The main goal of this study is to describe the general mixing behavior of raw materials by means of studying the scalar field pattern of total relative temperature and turbulent viscosity through CFD simulation, so that, the conditions of maximum biogas production can be precised for vast applicable fields.

Though anaerobic digestion is one of the simplest methods for biogas production, but still the amount of biogas yields from anaerobic digestion process is as high as expected. One of the vital reasons for such disturbance for sufficient biogas production is lack of improper mixing provided in the digester. Because, proper mixing of the slurry is very important for biogas production [I]. Since, anaerobic digestion is totally regulated by bacterial species, therefore, temperature plays very important role to accelerate the microbial activity. Besides, a desired level of P^H is also important to achieve successive amount of biogas from anaerobic digestion process.

Presently, models are becoming a more effective and powerful tools to represent the digesters behavior. As the digester is an opaque system, so it is impossible to optimize the digestion condition with open eyes. In such situation, computational modeling is very useful technique to understand the anaerobic digestion process by means of studying some selected parameters. In this investigation, the behavior of total relative temperature, effective viscosity and turbulent viscosity have been selected. By studying both effective viscosity and turbulent viscosity, the pattern of the interaction between the slurry layers

can be realized.

Even some modeling can be found by using fluid dynamic models for investigating anaerobic digestion performance. In some studies, the importance of mixing indicators such as retention time and the degree of mixing have been pointed out [II, III]. CFD can become more powerful if it can be done by suitable and convenient commercial software. In this study, a very beneficial commercial software, star CCM+ has been used. This software provides well-fitted turbulence models and numerical models with agreeable physics conditions. The laws governing fluid dynamics, solving equations by a geometric domain can be solved numerically by using this software. The basic flow equations used are the standard k-ε turbulence model, the Reynolds-Average Navier-Stokes, Viscous Fluxes in laminar flow and turbulent, and momentum equations were in discrete form. Besides, turbulence suppression was selected from viscous region box of the software. The magnitudes of temperature, effective viscosity and turbulence viscosity are calculated in a discrete manner at the nodes of a mesh, describing the flow geometry modeling.

Fig.1 Geometry of the mimic digester

At first step of developing the CFD modeling, the mimic geometric structure of an anaerobic digester was made. the replica of the biogas digester was built using AutoCAD 3D software, and then converted the file type lithography in order to be implemented in STAR CCM+ Software. For convenience of the simulation, the dimensions of the digester were changed. The height of the digester was 0.85m and the diameter of the digester was considered to be 0.35m. The inlet pipe for the insertion of the raw material and the biogas outlet was also made with suitable dimensions. Figure 1 is representing the schematic geometry of the digester.

Fig. 2(a) Sketch of the surface mesh

In the second step of modeling, the meshing was done with some selective meshing models. To make a suitable interconnection with CFD, both surface and volume mesher were considered. For surface mesher, surface remesher and surface wrapper remesher were selected. The conditions considered for volume meshing were: volume polyhedral mesher, and prism layer mesher. The surface mesh has generated 74950 cells and volume mesh has generated 426896 cells, 23364568 vertices and 2799367 faces. The attained mesh was very precise with cell size of 0.20 m it was

Fig. 2(b) Sketch of the volume mesh

appropriate for the dimensions of model. Figure 2(a, b) is presenting both of the surface and volume mesh during simulation.

CFD simulation inside the biogas digester was carried out with the help of the most well known standard k-ϵ model. This is the most widely used engineering turbulence model for industrial applications because it contains sub-models for compressibility, buoyancy, combustion, etc. Despite the fact that, the simulation was carried out for steady state, for better integrity and accuracy, governing equations are conferred here in unsteady state.

With the concept of Reynolds time-averaging and the rules defining its application, we turn our attention to the general conservation equations governing fluid flow and transport phenomena. The transport equations [IV] are mentioned below,

First, we consider incompressible fluids with constant properties. First, we consider incompressible fluids with constant properties. The equation of continuity is given by,

$$\frac{\partial \rho}{\partial t} + \nabla . \rho U = 0 \qquad (1)$$

Momentum equation is as follows,

$$\frac{\partial \rho U}{\partial t} + (\nabla . \rho UU) = -\nabla P + \nabla . \tau + \rho g \qquad (2)$$

Turbulent kinetic energy, k is given by,

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \quad (3)$$

Dissipation arisen due the turbulent flow is,

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_s}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1s}\frac{\varepsilon}{k}(G_k + C_{3s}G_b) - C_{2s}\rho\frac{\varepsilon^2}{k} + S_s \quad (4)$$

Relative total temperature was computed by means of the following governing equation,

$$\rho C_\rho \frac{DT}{Dt} = \nabla . \lambda_e \nabla T - \nabla . \sum_l \rho h_l (T) D_e \nabla m_l - \rho \sum_l \frac{Dm_l}{Dt} h_l (T) \qquad (5)$$

And the turbulent viscosity is given by,

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \qquad (6)$$

Besides, during specifying the boundary conditions for simulation, a classical velocity inlet boundary condition has been chosen. The biogas outlet was first defined as a pressure outlet; the rest of geometry was represented by boundaries conditions of wall. Physical conditions in the model are namely: 3D model, steady state. The physical properties of a continuous fluid with constant density, liquid was chosen as selected fluid flowing inside the digester. Segregated flow was being specified for simulation.

After specifying the conditions of the model, several iterations were implemented. Not only this, relative total temperature and turbulent viscosity digester were also plotted graphically. Figure 4 and Figure 5 are presenting the simulated results. The blue shaded region is presenting the lower value of the simulated output and red colored region is presenting the higher simulated values.

The optimum temperature range for obtaining maximum biogas after anaerobic digestion is from 299.15K to 303.15K (mesophilic temperature). From figure 4 it can be noticed that, most of the area of the scalar distribution pattern is covered with yellow and orange color. The reason of such color distribution is that, the temperature range is lying from 300.03K to 300.014K and this is an indicator to the mesophilic temperature range inside the digester.

Figure 5 is representing the simulated output for turbulent viscosity. From these figure it can be observed that, the value of turbulent viscosity has been reached a highest value and then it was found to be harmonically maintained. To obtain such result, a large number of iterations have been done. From these figures it also can be noticed that, turbulent viscosity was fluctuating within a range of 11-14Pa-s, with the highest value of 16Pa-s and the lowest value of 10Pa-s. This harmonic variation of turbulent viscosity is obtained due to the regular interaction between the fluid layers. Besides, the values of residuals after CFD simulation have been shown in Figure 6.

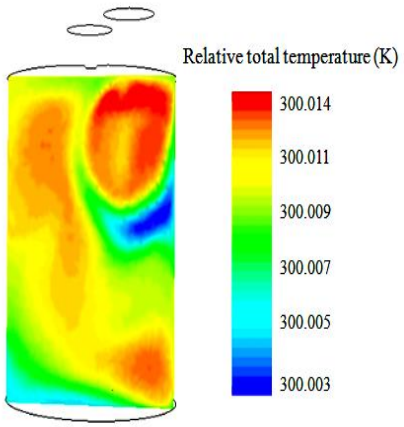


Fig. 4 Scalar field pattern for relative total temperature

In this paper, a modern and effective technique, CFD simulation was performed. The applied cease, boundary conditions and mathematical formulations envisioned the observed flow pattern in the digester. The main objective of this study is to make an acute observation on the hydrodynamic behavior of the raw materials inside a digester. To accomplish preferable output, a large number of iteration was performed. As CFD simulation is based on different numerical models and it complies on the basis of approximations. In such case, large number of iteration is advantageous for good result. In this paper, the iteration numbers are kept at wide range for the simulation in case of observing every parameter. There are several turbulence models to describe the hydrodynamic behavior of the materials in the digester, but our focus was to utilize standard k- ε turbulence model. The k-ε model has been designed specifically for planar shear layers and re-circulating flows. This model is the most widely used and validated turbulence model with applications ranging from industrial to environmental flows. The most important and critical step for CFD simulation is to make the finest meshing. Because the quality of meshing has the highest influence of the calculations for the solver to make sure whether it will converge and produce good result or diverge and giving error result. In this study the choice of meshing models was very well fitted, the base size of each cell

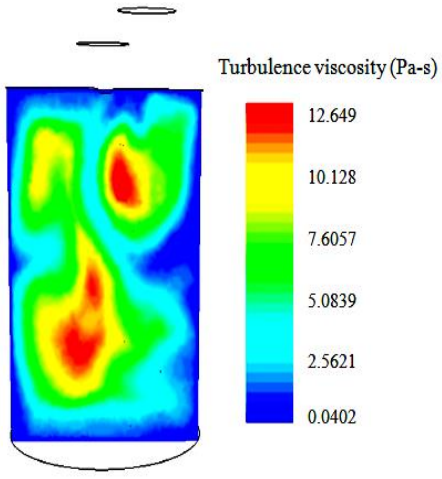


Fig. 4 Scalar field pattern for turbulent viscosity

was defined properly. The residual monitoring is also very useful tools to monitor the iteration steps whether it converge or diverge. The residual will be converge when the value is stable and not further down. Figure 9 is presenting the values of residuals after simulation. In this figure it can be monitored that, the values are taken within a large number of iteration and the values are very stable. This is a decent indication for the converging output of CFD simulation. The residual monitoring is also very useful tools to monitor the iteration steps whether it converge or diverge. The residual will be converge when the value is stable and not further down. Figure 9 is presenting the values of residuals after simulation. In this figure it can be monitored that, the values are taken within a large number of iteration and the values are very stable. This is a decent indication for the converging output of CFD simulation.

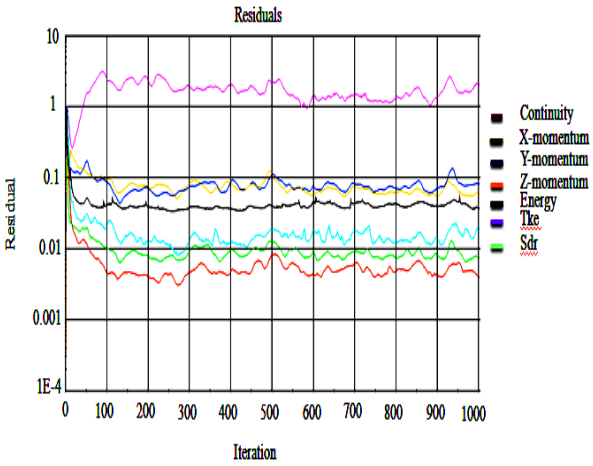


Fig. 6 Values of residuals after simulation

The main objective of this study is to understand the mixing behavior of sludge by means of the scalar field pattern of temperature and turbulent viscosity in order to produce maximum biogas. Accumulating the output of our work with the result of future appliances stated above, will be useful to design and implement a real anaerobic digester in medium and large scale, so that the energy crisis occurring in the modern era can be eliminated.

Author 1 Huqe Farhana received B.Sc. (Hons) and M.Sc. from department of Physics in University of Dhaka, Bangladesh. She is currently working as a PhD student at Advanced Mechanical System in Kumamoto University. Her research interest include biomass convertion by anaerobic digestion and application of computational fluid dynamics for designing convenient anaerobic digester

Experimental investigation on droplet impingement behavior on low-surface energy solids

Issei Tomimitsu[1], Yukihiro Yonemoto[2*], Yosuke Kawano[1], Tomoaki Kunugi[3]
[1]*Department of Mechanical and Mathematical Engineering, Kumamoto University*
[2]*Advanced Thermal and Fluid Energy System, Division of Industrial Fundamentals, Faculty of Advanced Science and Technology, Kumamoto University*
[3]*Department of Nuclear Engineering, Kyoto University*

*Corresponding author: Yukihiro Yonemoto
E-mail: yonemoto@mech.kumamoto-u.ac.jp
Keywords: Wettability, Droplet impingement, Fingering, Splashing

Abstract

Wetting behavior of a droplet on solid surfaces is observed in a variety of situation both industrial applications and daily life. For example, ink-jet printing and spray coating must be prevented droplet from breaking to gain the better finish condition. On the other hand, for fuel injection of the engine, burning splashed droplets will lead to improve the combustion efficiency. In either case, it is important for industrial application to understand the threshold criterion of droplet splashing behavior. The objective of the present study is to observe the droplet impingement behavior experimentally in detail in order to understand the criteria between the stable and the unstable conditions of droplet shape.

In the present study, impingement of water droplet on silicone rubber and polycarbonate substrates are investigated. From the experiment, three characteristic behaviors are mainly observed as shown in the Fig1. One is the deposition region (Fig. 1(a)). The stable shape of liquid film is observed in the spreading process. The more impact velocity increases, the more the droplet spreads over the solid surface. The other is fingering (Fig. 1(b)) and splashing behaviors (Fig. 1(c)). In Fig. 1(b), fingers are formed on the liquid film edge in radial direction. In Fig. 1(c), secondary droplets are ejected from the main droplet. In the fingering and splashing behaviors, instability of droplet shape after the impingement are observed. Finally, on the basis of the experimental results, we compare them with existing models for the splashing conditions.

(a) Deposition

(b) Fingering

(c) Splashing

Fig.1 Impingement behavior of a water droplet on Polycarbonate.

Issei Tomimitsu is 1st grade student of master course in the Department of Mechanical and Mathematical Engineering, Kumamoto University. He received the B.E. degree in Department of Mechanical Engineering from Kumamoto University, Japan, 2018. His research interests include fluid engineering and wettability.

Suspension Impact Acoustic Machine for Buildings

Tatsuya Hirayama, Saeko Tokuomi
Department of Mechanical and Mathematical Engineering, Kumamoto University

Corresponding author: Saeko Tokuomi
E-mail: stokuomi @st.kumamoto-u.ac.jp

Keywords: Impact acoustic method, suspension, Remote control, Building

Introduction

Inspection methods for concrete structures is usually undertaken using a hammering test. Typically in this kind of test, workers laboriously tap concrete surfaces with a hammer. This kind of work is neither economical in terms of time or money. So, I am in the process of developing a suspension testing machine that tests concrete walls, as shown in Fig. 1. Previously, the weakness of this machine was the noise coming from the fans, however, this was solved with 8 bladed fans as shown in Fig. 2.

Suspension Hammering Tester

The suspension hammering tester is rappelled down from the rooftop of buildings. The machine has four wheels, four impactors, two microphones, and four fans that apply pressure in order for the machine to adhere to concrete walls. The dimensions of the machine are 70 cm wide, 80 cm long and 18 cm thick. The weight the machine is 14.2 kg.

Fig. 1 Suspension tester.

Test Results

The noise of fans used in non-destructive testing methods such as drones is a serious problem. Similarly, I also had to address this problem as the machine includes fans near the microphones. The frequency of the impact sound on the test wall was 0.2 kHz, while the frequency of the fans was 0.55 kHz. Sometimes, there is a possibility that these frequencies will overlap.

Fig.2 8 bladed fan

The main frequency, f rises when either the rotational speed of the fans, n increase or the number of the blades, b increase. So, eight-bladed fans were trialed. The main frequency of these fans was increased from 0.55 kHz to 2.5 kHz, out of the impact frequency range. This resulted in more accurate detection of defects.

Tatsuya Hirayama received the B. E. degrees in mechanical engineering from Kumamoto University, Japan, in March of 2018. He is currently a student in the Department of Mechanical and Mathematical Engineering, Graduate School of Science and Technology Kumamoto University. His research interests include robotics, automobile and unmanned aircraft.

The design and The design and implementation of a vacuum suction micromouse

Li-Chun , Tsai
Southern Taiwan University of Science and Technology
Department of Electronic Engineering
Corresponding author: Li-Chun , Tsai
E-mail: ma630220@stust.edu.tw

Keywords: Mobile robot, micromouse, maze, flood-fill method, A algorithm*

Abstract

Micromouse is a mobile robot designed to navigate through an arbitrary maze. Micromouse competitions are a big challenge and opportunity for robotics research. They create student interest and inspire them to work dedicated on their projects. The working mouse must quickly roam around the maze and learn the route to the center, then travel from a designated maze corner to the center as rapidly as possible.

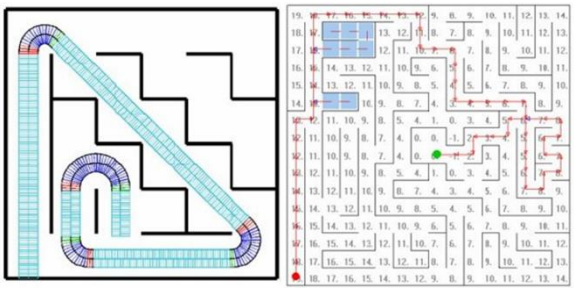


Fig. 1 Maze algorithm.

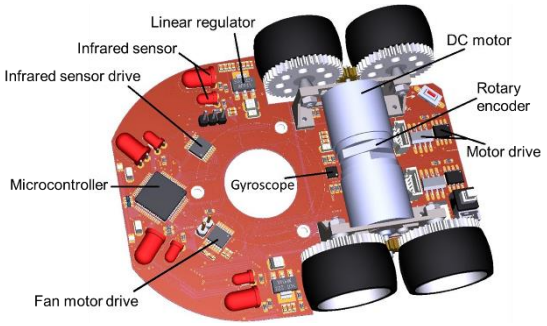


Fig. 2 Micromouse Hardware.

The Micromouse encompasses a vast range of engineering fields which can be divided into two categories: hardware and software. The hardware has been further subdivided into components of a more manageable size. The different hardware components are: power, sensors, control, and drive train. The power system consists of the battery pack and voltage regulation scheme in the circuit. The sensors are the means through which the Micromouse detects walls and traverses the maze with proper alignment in the center of a pathway. The drive train includes the motors and motor controllers, which produce the motion of the robot. Finally, the control unit is responsible for controlling each of the other components. Each of these parts in our project underwent a thorough process of design, analysis, and component selection.

Packing four copies of a (p,q)-graph

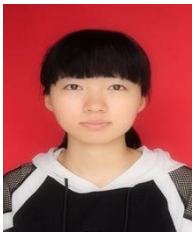
Yun Wang
School of Mathematics, Shandong University, Jinan 250100

Corresponding author: Jin Yan
E-mail: yanj@sdu.edu.cn

Keywords:packings, (p,q)-graph, k-placeable

Abstract

A graph G with order n is said to be k-placeable if there are k edge-disjoint copies of G in Kn. Let p, q be two integers. A (p, q)-graph is a graph of order p with q edges. There are two principle results in the paper. The first shows that all (n, n + 1)-graphs with minimum degree at least 2 and girth at least 9 are 4-placeable. The second characterizes (n, n − 1)-graphs with girth at least 9 which are 4-placeable.



Yun Wang is a second year Master of operations research student at Shandong University**.** Her research interests include packing and K-free graph.

Development of Training Machine for Hairpin Net Shot in Badminton

Ryo Iwamoto, Kazuya Mori
Department of Mechanical and Mathematical Engineering, Kumamoto University

Corresponding author: Kazuya Mori
E-mail: kmori@mech.kumamoto-u.ac.jp

Keywords: Badminton, Hairpin, Training machine,

Introduction

Badminton is becoming more and more popular in Japan. The number of badminton players in Japan is 7% of the total population. It has the twelfth highest participation rate after sports like baseball and table tennis [1]. Players need to master a variety of shots such as the clear shot, smash shot, drop shot, drive shot and the hairpin net shot. The hairpin net shot is a particular effective shot, however, requires high level of skill. To master this shot, it is necessary to repeatedly practice it over and over. Ideally, players should be able to practice this shot in personal and by themselves. So, I am developing a personal training machine for practicing the hair pin net shot as shown in Fig. 1.

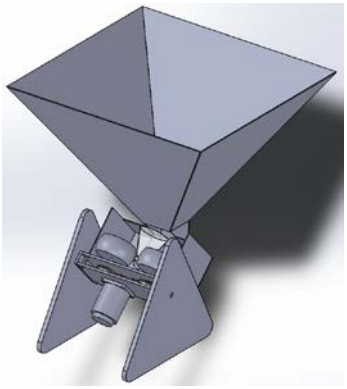


Fig. 1 Training machine.

Personal Hairpin Net Shot Training Machine

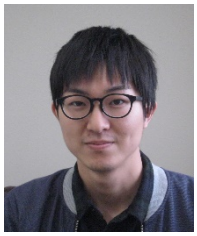
Fig. 1 shows the personal hairpin training machine which is being developed at Kumamoto University. After a player hits a hairpin net shot, the shuttle goes over the net and drops into a 50 cm by 50 cm basket. The shuttle funnels down into a launch device.

Fig. 2 shows the shuttle launch device which has two rotational wheels for launching the shuttles. The wheels are driven by a speed control motor (90 to 1450 rpm). The range of the launching speed is from 0.4 to 6 m/s. The machine can launch the shuttle at any set angle and at any set height over the net.

Fig.2 8 Launch device

Reference

[1] Ministry of internal affairs and communication, “Japanese sports statistics,” Statistics Topics No. 14, 2017.



Ryo Iwamoto was born November 2, 1995. He received the B. E. degrees in mechanical engineering from Kumamoto University, Japan, in 2018. He is currently a student in the Department of Mechanical and Mathematical Engineering, Graduate School of Science and Technology, Kumamoto University. His research interests include robotics, sport machine.

CFD Modeling of Gasification for Syngas Characterization from Palm Empty Fruit Bunch Biomass

Priyambodo Nur Ardi Nugroho[1,2], and Shuichi Torii[1]

1 Department of Advanced Mechanical System Engineering,
Kumamoto University, Kurokami, 2-39-1, Kumamoto 860-8555, Japan.
2 Politeknik Perkapalan Negeri Surabaya,
Jl. Teknik Kimia, Kampus ITS Sukolilo, Surabaya 60111, Indonesia

Corresponding author: Shuichi Torii
E-mail: torii@mech.kumamoto-u.ac.jp

Keywords: CFD, particle size, equivalence ratio, gasification parameters

Abstract

Biomass resource for bioenergy production is an indispensable alternative to fossil fuel energy and has been drawn attention. The natural form of biomass, which is bulky and inconvenient to handle, store, or transported efficiently, has become the primary idea of converting biomass. Gasification converts solid, liquid, or gaseous fossil or non-fossil fuels into useful gases or other chemicals. Characteristics of empty fruit bunch gasifier using Computational Fluid Dynamics (CFD) simulation model was discussed. Several essential gasification parameters were examined, such as syngas yield, syngas composition, Lower Heating Value (LHV), and Cold Gas Efficiency (CGE). Parameter performances were investigated with the variation of biomass particle size and Equivalence Ratio (ER). Two biomass particle sizes were 12 mm and 11 mm, while the equivalence ratio is spanning from 0.2 to 0.5. In simulation models, ER was achieved by varying air inlet rate.

Mass fraction of syngas composition, such as CO, CO2, CH4, and H2 was analyzed in conjunction with syngas yield, LHV, and CGE. As shown in Fig.1, the gasifier performance parameter (syngas yield, LHV syngas, CGE) have demonstrated an increasing trend as ER increase from 0.20 to 0.35 and reducing the biomass particle size. Maximum CGE reached with ER 0.35 and biomass particle size 11mm which is 51.32%, also, bigger biomass particle size shows 44.36% CGE at the same ER, which has been an improvement and significant input of this study. Optimal operating parameters were found to be ER 0.35, and finer biomass particle size would significantly affect gasification performance for a selection of proper operating condition.

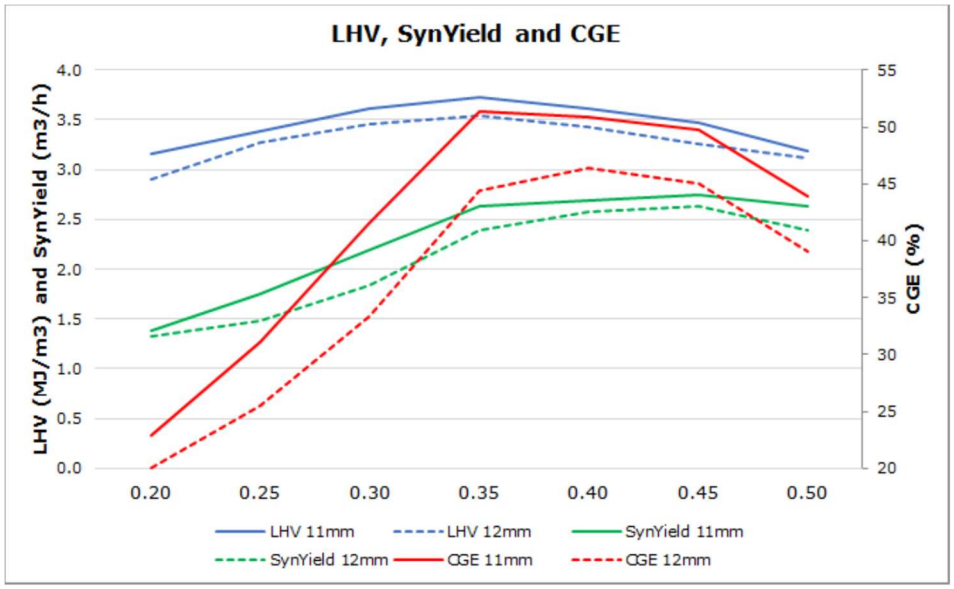


Fig. 1 Syngas LHV, Syngas Yield & CGE

Priyambodo Nur Ardi Nugroho received the B. E., and M. E. degrees in mechanical and industrial engineering from Institut Teknologi Sepuluh Nopember and Universitas Indonesia, in 2004 and 2010, respectively. He is currently pursuing doctoral degree in Advanced Mechanical System, Kumamoto University. His research interests focusing in renewable energy, particularly biomass gasification.