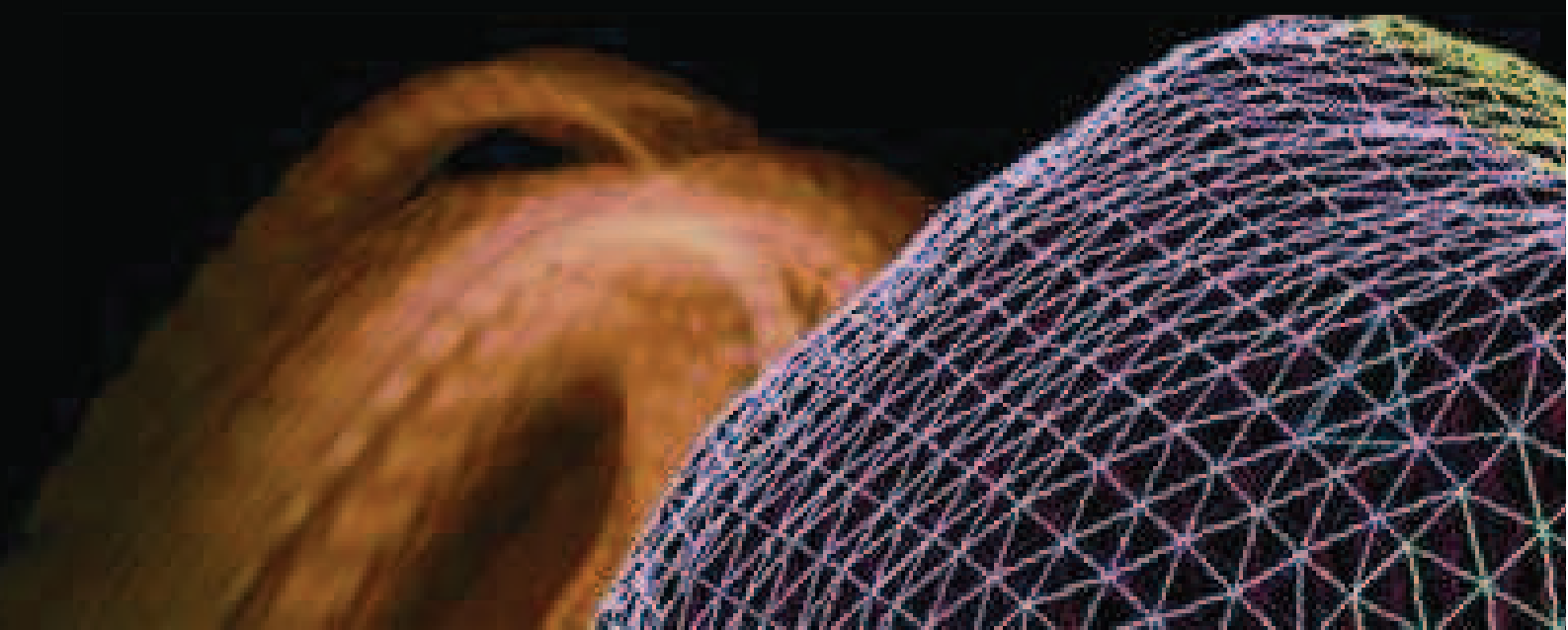
ISSN (Online) 2409-1669
ISSN (Print) 2409-7527

MULTIPHYSICS 2010

8 - 10 December 2010
Kumamoto, Japan



MULTIPHYSICS 2010

08-10 December 2010
Kumamoto, Japan

Conference Board

Chair

Moatamedi M, Narvik University College, Norway

Vice-chairs

Hokamoto K, Kumamoto University, Japan
Itoh S, Okinawa National College of Technology, Japan
Torii S, Kumamoto University, Japan

Scientific Committee

- Benson DJ, University of California, USA
- Chen S, Wuhan University, China
- Cross M, Swansea University, UK
- Elger G, Philips Technology GmbH , Germany
- Ewan BCR, University of Sheffield, UK
- Gabrys J, The Boeing Company, USA
- Giannopapa C, European Space Agency, Netherlands
- Itoh S, Okinawa National Collage of Technology, Japan
- Kayvantash K, CadLm, France
- Kwon Y W, Naval Postgraduate School, USA
- Moodie K, Health & Safety Laboratory, UK
- Oechsner A, University Technology, Malaysia
- Rahulan T, University of Salford, UK
- Ren Z, University of Maribor, Slovenia
- Schäfer M, Darmstadt University of Technology, Germany
- Shahrour I, University of Lille, France
- Tehrani A, Health and Safety Executive , UK
- Wakes S, University of Otago, New Zealand
- Zeguer T, Jaguar Land Rover, UK
- Zhong D, Tianjin University, China

General Information

Scope of Conference

Understanding real physics and performing multiphysics simulation are extremely important to analyse complex systems in order to better design and manufacture engineering products.

The objective of the conference is to share and explore findings on mathematical advances, numerical modelling and experimental validation of theoretical and practical systems in a wide range of applications.

The scope of the conference is to address the latest advances in theoretical developments, numerical modelling and industrial application which will promote the concept of simultaneous engineering. Typical combinations would involve a selection from subject disciplines such as Acoustics, Electrics, Explosives, Fire, Fluids, Magnetic, Soil, Structures and Thermodynamics.

Registration Pack – Collection Hours

Registration packs should be collected from the Registration Desk. Collection Hours are as follows:

Wednesday, 8th December	09:30-17:30
Thursday, 9th December	09:30-17:30
Friday, 10th December	09:30-14:00

Special Events

- Wednesday 18:00
  Welcome Reception

- Thursday 19:00
  Conference Banquet

Timing of Presentations

Each paper will be allocated 20 minutes. A good guide is 15 minutes for presentation with 5 minutes left for questions at the end.
Good timekeeping is essential, speakers are asked to keep strictly to 20 minutes per presentation.

Language

The official language of the conference is English.

Audiovisual

The lecture room will be preset with the following:
One laptop, one LCD projection and cables, one screen, and one microphone.
Delegates are requested to bring presentations on CD or memory stick.

Paper Publication

Selected papers are reviewed and published in 'The International Journal of Multiphysics'.

Sponsorship

Multiphysics 2010 is supported by Kumamoto University, Japan.
The Conference Board would like to thank the sponsors for their support.

Keynote Speaker

Professor Zoran Ren
University of Maribor, Slovenia

BIOGRAPHY

Professor Ren obtained his PhD degree in Advanced Computational Simulations from the School of Engineering at Swansea University. He has more than twenty years research experience in computational analysis of non-linear solid dynamics problems with emphasis on modelling of impact problems. In the past several years his expertise was also extended to fluid-structure interaction modelling and analysis. He is also active in research of mechanical engineering design and fracture mechanics problems. He has published a number of scientific papers and contributed to several books on the topics of his research. Professor Ren has been an invited lecturer at many scientific conferences and universities around the World. He has been a member of the organising committees for several conferences on computational mechanics and was also co organiser of the very first Multiphysics Conference, held in Maribor, Slovenia in year 2006.

His research work had a strong international connection, which includes collaboration through joint projects with the Swansea University, University of Glasgow, Delft University of Technology, Kettering University, University of Zagreb etc. His collaboration was recently extended also to Kumamoto University and Osaka University in Japan in the field of testing and simulation of advanced cellular materials.

Professor Ren has been awarded the prestigious Aleksander von Humboldt Research Fellowship in Germany, where he was researching ways of proper micromechanical modelling of fracture of crystalline solid materials at the Friedrich-Alexander University Erlangen-Nürnberg.

MULTIPHYSICS 2010

Programme

Time	Wednesday 8 Dec 10	Thursday 9 Dec 10	Friday 10 Dec 10
09:30 – 11:00	Registration	Session 2.1 *Fluids & Hydrodynamics*	Session 3.1 *Thermal Energy 1* *(Special Session)*
11:00 – 11:30	Coffee Break		
11:30 – 13:00	Keynote Address	Session 2.2 *Heat & Thermo-dynamics*	Session 3.2 *Thermal Energy 2* *(Special Session)*
13:00 – 14:00	Lunch		
14:00 – 15:30	Session 1.3 *Dynamic Modelling*	Session 2.3 *New Applications*	Session 3.3 *Food Processing*
15:30 – 16:00	Tea Break		
16:00 – 17:30	Session 1.4 *Modelling Advances*	Session 2.4 *Impacts & Explosions*	CLOSE
18:00	Welcome Reception		
19:00		Conference Banquet	

Full Programme

Wednesday 8 December 2010

09:30-11:30 **Registration**

11:30-13:00 **Keynote Address**

Chair: M Moatamedi, Narvik University College, Norway

Multiphysics Computational Simulations - challenges and opportunities
Z Ren, Maribor University, Slovenia

13:00-14:00 **Lunch**

Wednesday 8 December 2010

14:00-15:30 Session 1.3 - Dynamic Modelling
Chair: T Rahulan, University of Salford, UK

Dynamic Property of Aluminum Foam
Seiichi Irie, Graduate School of Science & Technology, Kumamoto University, Japan

Decomposition of Strain Energy in Cylindrical Shell Vibrations
Dr Basem Alzahabi, Dept of Mechanical Engineering, Kettering University, USA

Dynamic Axial Crush of Adhesive-bonded aluminum tubular structure
Minoru Yamashite, Centre for Advance Die Engineering & Tech, Gifu Uni, Japan. Toshio Hattori, Dept of Mech & System Eng, Gifu Uni, Japan

Numerical simulation of liquid sloshing using arbitrary Lagrangian-Eulerian level set method
Tadashi Watanabe, Centre for Computational Science & e-Systems, Japan Atomic Energy Agency, Tokai-mura, Naka-gun, Ibaraki-ken, Japan

15:30-16:00 Tea Break

Wednesday 8 December 2010

16:00-17:30 Session 1.4 - Modelling Advances
Chair : Z Ren, University of Maribor, Slovenia

Multiphysics Modelling of Atmospheric Ice Accrettion on Wind Turbines
Dr Muhammad Virk, Mr Matthew Homola, Prof Per J Nicklasson, Dept of Tehcnology, Narvik Uni College, Narvik Norway

A simple expression of coordinate transformation to simulate a number of optical devices
Shang-Ru Yu & Tungyang Chen, Dept of Civil Eng, National Cheng Kung Uni, Tainan, Taiwan

Multiphyscis approach applied to model properties of capacitive micromachined ultrasonic transducer
Adam Martowicz, Michal Manka, Tadeusz Uhl, AGH University of Science & Technology Dept of Robotics and Mechantornics Krakow, Poland

CFD-DEM Simulation of propagation of sound waves in a gas-solid fluidised medium
H.A. Khawaja, PhD Student, Department of Engineering, University of Cambridge, UK. D.S. Sultan, Department of Chemical Engineering, University of Cambrige, UK. S.A. Scott, Lecturer, Department of Engineering, University of Cambridge, UK

18:00 Welcome Reception

Thursday 9 December 2010

09:30-11:00 Session 2.1 – Fluids & Hydrodynamics

Chair: M S Virk, Narvik University College, Norway

CFD-DEM Simulation of reacting solid particles in a fluidised bed
H.A. Khawaja, PhD Student, Department of Engineering, University of Cambridge, UK. S.A. Scott, Lecturer, Department of Engineering, University of Cambridge, UK

Validation of a DEM Modeling of Gas-Solid Fluized Bed using the S-static in the State-Space Domain
M. Karimi, N Mostoufi, R Zarghami, R Sotudeh-Gharebagh. Multiphase Systems Research lab, Oil & Gas Centre of Excellence, School of Chemical Eng, Univerity of Tehran, Iran

Fluidized Bed Hydrodynamic Recognition Using Recurrence Plot
B Babaei, R Zarghami, R Sotudeh-Gharebagh, H.S. Kamal, N Mostoufi

Manufactruing of emulsion by underwater shock wave
T. Fujii (1) A Oda (2) A Takemoto (3) K Hokamoto (3) S Itoh (4)
1=Graduate School of Science & Technology, Kumamoto Univesity, 2=Human Resouce Center for Innovation, Kumamoto Uni. 3=Shock Wave & Condensed Matter Research Center, Kumamoto Uni
4=Okinawa National College of Technology, Japan

11-11:30 Coffee Break

Thursday 9 December 2010

11:30-13:00 Session 2.2 - Heat & Thermodynamics
Chair: B Alzahabi, Kettering University, USA

Thermal Wave Propagation Phenomena in a Thin Film Heated at Asymmetrical Wall Temperature
Shuichi Torii, Dept of Mech System Engineering Kumamoto Uni, Japan

Thermal Fluid Flow Transport Characteristics in confirmed channels with two-dimensional duel jet impringement
Caner Senkal & Shuichii Torii, Graduate School of Scient & Tech Mechanical System Eng, Kumamoto Uni, Japan

Basic Study on Explosive Evaporating Phenomena of Cryogenic Fluids by Direct Contacting Normal Temperature Fluids
Toshiaki Watanabe, Dept of Ocean Mech Eng, National Fisheries Univ. Japan. Ayumi Shiraishi, Hironori Maehara, Ayumi Takemoto, Shiguru Itoh

Irregular diamond-shaped column heating Vortex generated to discussion produced the phenomenon
Shuichi Torii Kumamoto and Lin-zijie, Kumamoto, Japan

13:00-14:00 Lunch

Thursday 9 December 2010

14:00-15:30 Session 2.3 - New Applications
Chair: T Watanabe, National Fisheries University, Japan

Energy Absorption Capacity of Trailer Under-ride Guard
Dr Basem Alzahabi, Dept of Mechanical Engineering, Kettering University, USA

Prediction of load-displacement curve for weld-bonded joints for dissimilar materials and thickness
Essam Al Bahkali, Jonny Herwan, Depatment of Mech Eng, College of Eng, King Saud Uni, Saudi Arabia

Coupled Gas Flow, Diffusion and Reaction in a ppb-level Sn-2-NiO Formaldehyde Sensor System
K Darcovich, J J Tunney, J L Dunford, L M Styles, G Xiao 1 and A Constant 2 National Research Council of Canada, Insti for Chemical Process and Environmental Tech, Ottawa, Ontario, Canada

Numerical evaluation of pressurizing conditions on explosive welding technique using reflected underwater shock wave
Akihisa MORI, Kazumasa Shiramoto and Marahiro FUJITA, Sojo Uni, Kumamoto, Japan

15:30-16:00 Tea Break

Thursday 9 December 2010

16:00-17:30 Session 2.4 - Impact and Explosions
Chair: S Itoh, Okinawa National College of Technology, Japan

Research on initiation sensitivity of solid explosive
Mashahiko Otuka, Asahikasei Chemical Corpt, Hideki Hamasima, Kumamoto Indust Research Inst. Kazuyuki Hokamoto, Shock Wave & Condenses Matter Research Center, Kumamto Uni, Sigeru Itoh, Okinawa National College of Tech, Japan

Impact of explosive to civil and underground structures
Dolezel, V. Universita Pardubice, Czech Republic, Prochazka, P, Associal of Czech Civil Engineering, Prague, Czec Rep.

Influence of the fire in underground structures
Dolezel, V. Universita Pardubice, Czech Republic, Prochazka, P, Associal of Czech Civil Engineering, prague, Czec Rep.

Fluid-Structure Interaction analysis of a steel pipe subjected to internal blas loading
Tei Saburi, Shiro Kubota & Yuji Ogata, National Institute of AIST, Onogawa, Japan

19:00 Conference Banquet

Friday 10 December 2010

09:30-11:00 Session 3.1 - Thermal Energy 1
Chair: S Torii, Kumamoto University, Japan

Investigation of Size Effects to the Mixing Performance on the X-shaped Micro-Channels
Shu-Min Tu, Kumamoto University, Japan

THERMAL FLUID FLOW TRANSPORT CHARACTERISTICS IN CONFINED CHANNELS WITH TWO-DIMENSIONAL DUAL JET IMPINGEMENT
C. Senkal, Kumamoto University, Japan

Development of New Model Combustor for Biomass
Masato Urashima, Kumamoto University, Japan

A Basic Research of Separation and Collecting Solid Materials from Solid-Liquid
Koichi Shimamoto, Kumamoto University, Japan

11:00-11:30 Coffee Break

Friday 10 December 2010

11:30-13:00 Session 3.2 - Thermal Energy 2
Chair: S Torii, Kumamoto University, Japan

Response Examination to Make Micro-pump with the Aid of Peltier Devices and Heat Deformation Material
Yasuhito Takakura, Kumamoto University, Japan

Transport phenomenon Vortex behind irregular diamond-shaped column heated under constant heat flux
Lin-zijie, Kumamoto University, Japan

Effect of Plate Shape on Thermal-Fluid Flow Characteristics in Plate Heat Exchanger
Keita Izumi, Kumamoto University, Japan

New Simulation technique of the LSC Jet
Hitoshi Miyoshi, Impact Eng Lab, Japan

13:00-14:00 Lunch

Friday 10 December 2010

14:00-15:30 Session 3.3 - Food Processing
Chair: K Hokamoto, Kumamoto University, Japan

The design of rice powder production vessel and the pulverization of the rice using numerical simulation
Manabu Shibuta, Graduate School of Science & Technology, Kumamoto Uni. Japan

The Basic Research for Pulverization of Rice Using Underwater Shock Wave by Electric Discharge
Makoto Ide, Graduate School of Science & Technology, Kumamoto University, Japan

On particle size distribution and the pressure condition of the rice powder manufactured from the instantaneous high pressure
Takemoto, Ayumi, Kudou Yasufumi, Mimaki Nami, Shigeru Itoh, Japan

Evaluation using numberical simulation of bubble pulsation in food processing using underwater shock wave
Hideki Hamashima, Naoyuki Wada, Manabu Shibuta, Kazuyuki Hokamoto & Shigeru Itoh Kumamoto Industrial Research Inst. Japan

16:00 CLOSE OF CONFERENCE

Keynote Address

WEDNESDAY 08 DECEMBER 2010
11:30 – 13:00

CHAIR

M Moatamedi
Narvik University College
Norway

Wednesday, 08 December 2010
11:30-13:00

Keynote Address

Multiphysics Computational Simulations - challenges and opportunities

Professor Zoran Ren
University of Maribor
Slovenia

Computational modelling of complex coupled problems in structural, fluid, thermal and electromagnetic applications has received much deserved attention in recent years, both in research and development of underlying computational procedures and their application for solving real-life scientific and applied problems. This increase of interest in multiphysics simulations is mainly due to availability of cost-effective high-performance parallel computing on dedicated personal computer clusters or more recently the onset of distributer or "cloud" computing. More complex problems can now be solved at much higher resolution in a reasonable computing time. This holds true also for computational simulations of spatial and temporal multi-scale problems, multidisciplinary and multilevel optimisation problems etc., which all require significant computational resources.
Multiphysics computational simulations are important to investigate complex problems in nature because in its systematic approach helps to identify and understand importance of certain parameters used to define the underlying physical relationships. Further development of computational tools for multiphysics simulations will undoubtedly play major role in future rational design of novel materials with desired properties in next generation nano electronic devices, alternative energy materials, life science etc. These in turn will further increase computing capabilities, in time maybe even beyond our needs.
The keynote address will focus on some present challenges and future opportunities of multiphysics computational simulations as perceived by the author.

SESSION 1.3

DYNAMIC MODELLING

WEDNESDAY 08 DECEMBER 2010
14:00-15:30

CHAIR
T Rahulan
University of Salford
United Kingdom

Dynamic Property of Aluminum Foam

Corresponding Author
Seiichi Irie
seiichi.1215@gmail.com

Authors
Seiichi Irie, Graduate School of Science and Technology, Kumamoto University, Japan
Toshihiko Okano, Shigeru Tanaka, Matej VESENJAK, Zoran REN, Kazuyuki Hokamoto, Shigeru Itoh

Aluminum in the foam of metallic foam is in the early stage of industrialization.
It has various beneficial characteristics such as being lightweight, heat resistance, and an electromagnetic radiation shield. Therefore, the use of aluminum foam is expected to reduce the weight of equipment for transportation such as the car, trains, and aircraft. The use as energy absorption material is examined.
Moreover aluminum foam can absorb the shock wave, and decrease the shock of the blast.
Many researchers have reported about aluminum foam, but only a little information is available for high strain rates (103s-1 or more). Therefore, the aluminum foam at high strain rates hasn't been not characterized yet.
The purpose in this research is to evaluate the behavior of the aluminum form in the high-strain rate. In this paper, the collision test on high strain rate of the aluminum foam is investigated. After experiment, the numerical analysis model will be made.
In this experiment, a powder gun was used to generate the high strain rate in aluminum foam. In-situ PVDF gauges were used for measuring pressure and the length of effectiveness that acts on the aluminum foam. The aluminum foam was accelerated to about 400m/s and 200m/s from deflagration of single component powder and the foam were made to collide with the PVDF gauge. The high strain rate deformation of the aluminum form was measured at two collision speeds. As for the result of both speeds, pressure was observed to go up rapidly when about 70% was compressed. From this result, it is understood that complete crush of the cell is caused when the relative volume is about 70%. In the next stage, this data will be compared with the numerical analysis.

Keywords
Aluminum foam, Powder gun, high strain rate

Decomposition of Strain Energy in Cylindrical Shell Vibrations

Corresponding Author
Basem Alzahabi
balzahab@kettering.edu

Author
Basem Alzahabi
Department of Mechanical Engineering
Kettering University, USA

Cylindrical Shells are widely used structures in many structural applications, such as offshore structures, liquid storage tanks, submarine, and airplane hulls and all are required to operate in a dynamic environment that varies in its severity. Therefore, investigating the dynamic characteristics of cylindrical shells is very critical in developing a strategy for modal vibration control. Reduction of vibration amplitudes and therefore sound radiation may be determined by considering operational scenarios and modal characteristics.
In cylindrical shells, mode shapes associated with each natural frequency are typically a combination of Radial, Longitudinal, and Circumferential modes. And while the lowest natural frequency corresponds to the lowest level of strain energy, it does not necessarily correspond to the lowest wave index. In fact, the natural frequencies do not fall in ascending order of the wave index in cylindrical shells. The total strain energy is the combination of membrane and bending strain energies. The ratio of membrane strain energy to the total strain energy is high for modes with simple modal patterns and decrease toward zero as the number of nodal (n) lines increase.
In this paper, a comprehensive study of the modal characteristics of a shear diaphragm supported thin circular cylindrical shell is presented. Analytical expressions for the equations of motion are derived using Donnell-Mushtari shell theory for uniform circular cylindrical shells. A comparative study of the natural frequencies and strain energy density for the corresponding mode shapes of uniform circular cylindrical shells using the analytical expressions and the Finite Element Analysis is performed.
Then an experimental modal analysis of the scaled model was performed to obtain the modal characteristics of the cylindrical shell between 0 and 800 Hz. The experimental data were correlated with those results obtained analytically and numerically using the finite element analysis. The experimental modal analysis of the scaled model is then repeated utilizing strain gauges to decouple the strain energy components. The results were found to be in excellent agreement.

Keywords
Cylindrical Shell Vibrations, Strain Energy

Dynamic Axial Crush of Adhesive-Bonded Aluminum Tubular Structure

Corresponding Author
Minoru Yamashita
minoruy@gifu-u.ac.jp

Authors
Minoru Yamashita: Center for Advanced Die Engineering and Technology, Gifu University, Gifu, 501-1193, Japan
Toshio Hattori: Department of Mechanical and Systems Engineering, Gifu University, Gifu, 501-1193, Japan

The tubular structures with hat-shaped cross-section were axially compressed under the dynamic condition. Two kinds of structures were compressed. Type I structure consisted of a hat part formed with V-bending and a flat plate. Type II consisted of two similar hat parts. The parts were bonded using an epoxide adhesive at the flange portion. The sheet materials were a pure aluminum A1050 and its alloy A5052, whose thickness was 1 mm. The impact velocity was 10 m/s, where a drop-hammer was used. The separation behavior at the bonded flange was highly visible for A5052 material, which is harder than A1050, in case of Type I structure. As for the A5052 material, the flange separation was more suppressed for the Type II structure than for Type I. This may be attributed to the geometrical symmetric property in cross-section with respect to the bonding plane, which avoids causing excessive exfoliating or tensile force at the bonding layer. Because the intrinsic wavy deformation pattern by compression is similar for both parts in Type II structure. The numerical simulation of axial compression of the adhesive-bonded tubular structure was also performed using the dynamic explicit finite element method. The separation behavior of the bonded flange was considered. Four-node shell elements were used for modeling the aluminum sheet. The bonding layer was modeled with hexahedron elements, where the number of division of the layer was single, because the computation was conducted as a trial. The mechanical properties of the adhesive were assumed mainly based on the shear bonding strength. The computational result demonstrates that the flange separation was more conspicuous for the A5052 material than for A1050, which coincides with the experimental result for the Type I structure. The crush strength during collapse was also well predicted.

Keywords
Dynamic deformation, Axial compression, Adhesive-bonded structure, Computation

Numerical simulation of liquid sloshing using arbitrary Lagrangian-Eulerian level set method

Corresponding Author
Tadashi Watanabe
watanabe.tadashi66@jaea.go.jp

Author
Tadashi Watanabe,
Centre for Computational Science and e-Systems, Japan Atomic Energy Agency,
Tokai-mura, Naka-gun, Ibaraki-ken, 319-1195, Japan

Oscillating two-phase flows with free surface are seen widely in engineering fields, and predictions of fluid phenomena with complicated surface motion are of practical importance, since such fluid motion may result in large pressure impact on structures. Liquid sloshing has been studied both experimentally and analytically in relation to sea transport of oil, fuel of spacecraft, seismic response of liquid metal reactors, and so on.

In this study, liquid sloshing in an oscillating tank is simulated numerically as a fluid-structure interaction problem. A stratified two-phase flow is contained in a rectangular tank, and the tank is set in oscillatory motion. The motion of the tank is modeled using the arbitrary Lagrangian-Eulerian method, where the computational grid points are moved with the velocity of the tank. Two-phase flow fields are solved using the level set method. In the level set method, the level set function, which is the distance function from the two-phase interface, is calculated by solving the transport equation using the flow velocities. Both the liquid phase and gas phase flow fields with the free surface motion induced by the oscillating tank are obtained by this ALE level set coupled method.
It is shown by comparing the simulation results with the existing experimental results that the sloshing behavior of the free surface is predicted well by the present method. The simulation results are also compared with the case using the body force, in which the body force term is included in the fluid equations and the grid points are not moved. The difference between the moving grid method and the body force method is made clear, and it is shown for small amplitude sloshing that the results with the body force are almost the same as the results with moving grid points.

Keywords
Sloshing, two-phase flow, free surface, oscillation, fluid-structure interaction

SESSION 1.4

MODELLING ADVANCES

WEDNESDAY 08 DECEMBER 2010
16:00-17:30

CHAIR
Z Ren
University of Maribor
Slovenia

Multiphysics Modelling of Atmospheric Ice Accretion on Wind Turbines

Corresponding Author
Dr Muhammad Shakeel Virk
msv@hin.no

Authors
Dr Muhammad S. Virk
Mr Matthew C. Homola
Prof Per J. Nicklasson
Department of Technology
Narvik University College, 8505 Narvik, Norway

Wind energy is a widely accepted source of power available everywhere in the world, which is not depleted over time. Most northern regions of the world like Arctic have good wind resources, but atmospheric icing on wind turbines has been recognized as a hindrance to the development of the wind power in these regions, where the uncertainty surrounding the effect of icing on energy production may prevent otherwise good wind resources from being utilized. Ice accretion on the wind turbine blades is caused by the impingement of super cooled water droplets. Most of these liquid water droplets freeze immediately upon impact due to rapid heat dissipation leading to ice accretion. Multiphysics modelling of atmospheric ice accretion on the wind turbine blades involves the calculation of droplet trajectories, interaction of air and droplets, droplet impingement efficiency and phase change from liquid to solid after impingement. Rate and shape of the accreted ice depends upon many variables such as point of operation, the geometry of wind turbine blade, relative wind velocity, temperature, droplet diameter and the liquid water content .
Computational fluid dynamics (CFD) based multiphase numerical analyses were carried out to understand the effect of different operating and geometric parameters variation on atmospheric ice growth on wind turbines. Results showed that an increase in blade profile size and decrease in the geometric twist angle reduces the atmospheric ice accretion, both in terms of local mass and ice thickness . An increase in the ice growth with increasing droplet size was observed; and change in atmospheric temperature significantly affects the shape of accreted ice. Streamlined ice shapes were observed for low temperatures, whereas horn shape ice accretions were found at higher temperatures . Results showed a decrease in power production for a simulated 5 MW wind turbine of approx 28% due to atmospheric ice accretion.

Keywords
Wind turbine, Atmospheric icing, CFD, Blade geometry, Temperature

A simple expression of coordinate transformation to simulate a number of optical devices

Corresponding Author
T. Chen
tchen@mail.ncku.tw

Authors
Shang-Ru Yu and Tungyang Chen
Department of Civil Engineering, National Cheng Kung University, Tainan 70101, Taiwan

We propose a simple expression of coordinate transformation that is capable to simulate a variety of optical devices with different functionalities in a unified manner. The expression involves one matrix function and one scalar vector, in which the variable involved in the matrix function is relevant to the geometric configurations of the device, while the vector governs the functionality of the device. We show that, with different choices of the parameters, the coordinate transformation can simulate a number of different devices, including invisibility cloaks, field rotators, concentrators, superscatterers, and translators, all within one framework. In addition to the conventional descriptions based on Cartesian or cylindrical coordinates, the transformation formula also applies to other curvilinear orthogonal coordinates. We demonstrate some numerical simulations for selected examples based on finite element calculations. This framework provides new perspectives to interpret the devices based on transformation optics.

Multiphysics approach applied to model properties of capacitive micromachined ultrasonic transducer

Corresponding Author
Adam Martowicz
adam.martowicz@agh.edu.pl

Author
Adam Martowicz, Michal Manka, Tadeusz Uhl
AGH University of Science and Technology
Department of Robotics and Mechatronics
Al. Mickiewicza 30, 30-059 Krakow, Poland

Capacitive micromachined ultrasonic transducers (CMUT) compose a class of microelectromechanical systems (MEMS) used for imaging technique and testing. CMUT consists of electrostatically actuated micron-scale membrane suspended over a silicon substrate. Alternating voltage generated in integrated electronic circuit is applied to power electrodes and makes the membrane vibrate. Hence the ultrasonic wave can be generated and then emitted to monitored object. A number of uniformly distributed CMUT cells can build a 1-D or 2-D array ultrasonic sensor used to speed-up the measurement process as well as to direct propagating wave. There are known applications of CMUT technology for medical diagnosis as well as for non-destructive testing performed to assess the health of monitored mechanical structures. Diagnostic information on the state of monitored object is derived from the parameters of reflected ultrasonic wave, e.g. time of flight, phase shift, amplitude. Larger and larger application area of CMUT encourages efforts to improve its performance and increase the quality of design.
Specificity of physical phenomena present in CMUT determines the necessity of multiphysics approach. In a general case there should be considered the following phenomena: structural dynamics to model basic mechanical properties, electrostatics for the membrane actuation, fluid-structure interaction to introduce the effect of fluid squeezing present in the area between membrane and substrate, and acoustics to model the propagation of ultrasonic wave in surrounding medium. The work deals with the multiphysics modelling of exemplary CMUT. Parameterized FE model is applied to study the influence of selected geometry and material properties on properties of CMUT. The results of carried out sensitivity analysis are discussed in the context of possible improvements of the design.
The work was supported by the Polish Grant no. N N503 141236, which deals with the Method of multiobjective optimization of the construction of microdevices considering technological uncertainties.

Keywords
Microelectromechanical systems, capacitive micromachined ultrasonic transducer, finite element method, imaging technique

CFD-DEM simulation of propagation of sound waves in a gas-solid fluidised medium

Corresponding Author
Hassan Abbas Khawaja
hak23@cam.ac.uk

Authors
H.A. Khawaja, PhD Student, Department of Engineering, University of Cambridge, UK
S.A. Scott, Lecturer, Department of Engineering, University of Cambridge, UK

In this work, speed of sound in 2 phase mixture has been explored using CFD-DEM (Computational Fluid Dynamcis Discrete Element Modelling). In this method volume averaged Navier Stokes, continuity and energy equations are solved for fluid. Particles are simulated as individual entities; their behaviour is captured by Newtons laws of motion and classical contact mechanics. Particle-fluid interaction is captured using drag laws given in literature.
The speed of sound in a medium depends on physical properties. It has been found experimentally that speed of sound drops significantly in 2 phase mixture of fluidized particles because of its increased density relative to gas while maintaining its compressibility. Due to the high rate of heat transfer within 2 phase medium as given in Roy, Davidson and Tuponogov (1990), it has been assumed that the fluidised gas-particle medium is isothermal.
The similar phenomenon has been tried to be captured using CFD-DEM numerical simulation. Two different methods have been used to examine this phenomenon in CFD-DEM simulations. Firstly, a disturbance is introduced and fundamental frequency in the medium is noted to measure the speed of sound for e.g. organ pipe. Secondly, by creating fluid bubbles in the fluidised medium, pressure fluctuations are created which can be tracked to establish their propagation speed. It has been found that speed of sound is in agreement with the relationship given in Roy, Davidson and Tuponogov (1990). Their assumption that the system is isothermal appears to be valid.
Reference:
Roy R., Davidson J. F., Tupogonov V. G., The velocity of sound in fluidized beds, Chemical Engineering Science, 1990, 45, 11, 3233-3245

Keywords
CFD, DEM, 2 phase mixture, speed of sound, fluidised particles

SESSION 2.1

FLUIDS AND HYDRODYNAMICS

THURSDAY 09 DECEMBER 2010
09:30-11:00

CHAIR
M S Virk
Narvik University College
Norway

CFD-DEM Simulation of reacting solid particles in a fluidised bed

Corresponding Author
Hassan Abbas Khawaja
hak23@cam.ac.uk

Authors
H.A. Khawaja, PhD Student, Department of Engineering, University of Cambridge, UK
D. S. Sultan, Department of Chemical Engineering, University of Cambridge, UK
S.A. Scott, Lecturer, Department of Engineering, University of Cambridge, UK

In this work, the interactions between chemical reaction and fluid mechanics are examined in a 2 phase medium using CFD-DEM (Computational Fluid Dynamics Discrete Element Modelling) to model a fluidized bed. In this method, volume averaged Navier Stokes, continuity, energy and species concentration equations are solved for multi-fluid flows. The particles are simulated as individual entities; their behaviour is captured by Newtons laws of motion and classical contact mechanics. Particle-fluid interaction is captured using drag laws given in literature.
Equations have been included to model the reaction of particles and the transfer of chemical species, energy and momentum to the fluid phase.
The developed code is capable of handling various types of solid particles-gas chemical reactions. For a test case, the carbonation and decomposition of Calcium Oxide particles in a fluidized bed will be studied. This reaction is strongly dependent on local temperature and partial pressure of Carbon Dioxide because of its reversible nature. Results of the simulation will be compared with analogous experiments in which Calcium Oxide has been repeatedly carbonated and calcined in a small laboratory scale fluidized bed.

Keywords
CFD, DEM, chemical reactions, Calcium Carbonate

Validation of a DEM Modeling of Gas-Solid Fluidized Bed using the S-statistic in the State-Space Domain

Corresponding Author
Reza Zarghami
rzarghami@ut.ac.ir

Authors
M. Karimi, N.Mostoufi, R. Zarghami, R. Sotudeh-Gharebagh
Multiphase Systems Research Lab., Oil and Gas Centre of Excellence, School of Chemical Engineering, University of Tehran, P.O. Box 11155/4563, Tehran, Iran

A reliable method was developed to validate results of a gas-solid bubble fluidized bed model with discrete element method (DEM) through comparison of corresponding pressure fluctuations experimental data. Attractors of two independent pressure signals, evaluation series of DEM model and reference time series of measured pressure signals, were compared in the state-space domain using the S-statistic. Comparison between two reconstructed attractors of evaluation and reference series was performed based on the null hypothesis. The null hypothesis that the evaluation and reference time series originate from the same dynamic sources is rejected if the two series significantly differ. To prove the power of the method, the S-statistic was estimated for obtained experimental data under the same operating conditions. In addition, experimental and model pressure fluctuations were decomposed into 9 levels using wavelet transform to study the validity of the model in a broad range of frequencies. Results indicated that the model results were consistent with experiments.

Keywords
S-statistic; Null-hypothesis; Discrete wavelet transform; Discrete element method (DEM); Fluidized bed, State-Space

Fluidized Bed Hydrodynamic Recognition Using Recurrence Plot

Corresponding Author
Reza Zarghami
rzarghami@ut.ac.ir

Authors
B. Babaei, R. Zarghami, R. Sotudeh-Gharebagh, H. S. Kamal, N. Mostoufi

The main purpose of this work is to introduce recurrence plot, an easy statistical method, to visualize and recognize fluidized bed hydrodynamic. Recurrence plot of pressure time series of fluidized bed has been plotted at different superficial gas velocities. There were four repeated patterns in all RPs; short diagonal lines, small bold areas, stretched white bands and quasi-square patterns. Hydrodynamic of the bed can be well explained using these patterns. Short diagonal lines indicate complexity of the fluidized bed hydrodynamic. In the other words, they show that hydrodynamic behavior of the bed is predictable only for short times. The size of bold areas is small at high velocities. This shows that the macro structures within the bed are more dominant at high velocities. When gas velocity is increased the white bands become wider. This confirms that at high gas velocities large bubbles produce greater pressure amplitudes. Quasi-square patterns represent various origins of pressure fluctuations such as bubble eruption and coalescence and particles interaction. The size of quasi-square patterns is increased at high velocities due to increase in bubble size. Macro, mezzo and micro structures can be easily redefined in terms of the size of these patterns.

Keywords
Fluidized Bed Hydrodynamic; Recurrence Plot; Recurrence Quantification Analysis

Manufacturing of emulsion by underwater shock wave

Corresponding Author
Takeshi Fujii
takeshi@shock.smrc.kumamoto-u.ac.jp

Authors
T. Fujii (1), A.Oda(2), A. Takemoto(3), K. Hokamoto(3) and S. Itoh (4)
1. Graduate School of Science and Technology, Kumamoto University
2-39-1 Kurokami Kumamoto, 860-0862 JAPAN
2 .Human Resource Center for Innovation, Kumamoto University
3. Shock Wave and Condensed Matter Research Center, Kumamoto University
4. Okinawa National College of Technology

The emulsification is the technology that very important and indispensable for recent industry. This technology is used in many fields. For example, it is foods (Dressings, Sauces, and Dairy products), chemistry and materials (Paints, Cosmetics, Fragrances, and Explosives), fuel (Emulsion fuel) etc...From this, the emulsification can be said to be the important technology and essential to our life. However, there are some problems in present emulsification technology. The main things are an emulsifier, emulsion stability and churning time in those problems. Emulsifier is a factor in rising of products cost and addition unnecessary properties. But, emulsion stability is not provided if emulsifier is not used. In addition, churning takes long time. The shock wave can solve these problems. The purpose of this research is cutting back on consumption of emulsifier and shortening churning time by using shock wave.
This experiment used under water shock wave. Detonating fuse of the pyrotechnic was used as shock wave outbreak source, and the #6 electric detonator was used for initiation. Sample was made of enclosing oil and water within polycarbonate bottle. The pressure of the water shock wave to on the sample can be regulated by distance with detonating fuse and the sample by changing it. The processed sample was measured particle size distribution and monitored about 1 month. In this way, the sample was determined stability. As a result, it?fs recognized that these samples have good stability.

Keywords
Emulsion, Underwater shock wave, Explosive, Food, Fuel

SESSION 2.2

HEAT AND THERMODYNAMICS

THURSDAY 09 DECEMBER 2010
11:30 - 13:00

CHAIR
B Alzahabi
Kettering University
USA

Thermal Wave Propagation Phenomena in a Thin Film Heated at Asymmetrical Wall Temperature

Corresponding Author
Shuichi TORII
torii@mech.kumamoto-u.ac.jp

Authors
Department of Mechanical System Engineering,
Kumamoto University

A numerical study is performed to investigate thermal wave propagation in a very thin film subjected to an asymmetrical temperature change on both sides. The non-Fourier, hyperbolic heat conduction equation is solved using a numerical technique based on MacCormak's predictor-corrector scheme. Consideration is given to the time history of thermal wave behavior before and after asymmetrical collision of wave fronts from two sides of a film. It is disclosed that in transient heat conduction, a heat pulse is transported as a wave in the film, and that non-Fourier heat conduction is extremely significant with certain range of film thickness and time. That is, sudden heating on both sides of the extremely thin film causes temperature overshoot within a very short period of time.

Keywords
Thermal Shock Wave, Non-Fourier Low, MacCormak's pedictor-corrector method

Thermal fluid flow transport characteristics in confined channels with two-dimensional dual jet impingement

Corresponding Author
Caner SENKAL
casenkal@yahoo.com

Authors
Caner SENKAL (Graduate School of Science and Technology Mechanical System Engineering Kumamoto University Kumamoto City, Japan)
Shuichii TORII (Graduate School of Science and Technology Mechanical System Engineering Kumamoto University Kumamoto City, Japan)

The flow and heat transfer characteristics of laminar dual circular jet impinging on a heating plate with inclined confinement surface has been investigated numerically.Governing equations in steady state are solved by a control volume based finite-difference method. The simulations have been carried out for reynolds number (250≤Re≤1000), the angle of inclination of the confined upper wall (0 ≤ θ ≤ 10), circular jet to annular jet velocity ratio (0≤VR≤2) and jet to target plate distances between 2D and 8D where D is the outer diameter of dual jet.SIMPLE algorithm was used to obtain velocity and temperature fields. Hybrid difference scheme is adopted for the discretized terms in the governing equations. The discretised equations are solved iteratively using the tridiagonal matrix algorithm line solver. Heat transfer performance along the heated wall is amplified with an increase in the velocity ratio and the reynolds number.On the contrary,a substantial reduction in the heat transfer rate, for VR=0.0, occurs in the stagnation zone, because the absence of the inner nozzle injection causes the recirculation in the corres-ponding region. The heat transfer rate in the stagnation zone is attenuated by increasing the jet nozzle to impinging plate distance. In particular, the effect of the inclination angle in the down-stream region,especially at the vicinity of outlet, is major then other effects Nusselt number distribution on the impingement plate is affected by inclined upper wall because inclination of the wall accelerates the exhaust flow. The streamwise reduction in the heat transfer rate for θ=0.0 is suppressed by the presence of the inclined confinement surface and its value is intensified by the inclination angle.

Keywords
Jet impingement; electronics cooling; heat transfer ; annular jet

Basic Study on Explosive Evaporating Phenomena of Cryogenic Fluids by Direct Contacting Normal Temperature Fluids

Corresponding Author
Toshiaki WATANABE
watanabe@fish-u.ac.jp

Authors
Toshiaki WATANABE: Dept. of Ocean Mech. Eng., National Fisheries Univ., Japan
Ayumi SHIRAISHI: Undergraduate school of Dept. of Ocean Mech. Eng., National Fisheries Univ., Japan
Hironori MAEHARA: Graduate school of Science and Technology, Kumamoto Univ., Japan
Ayumi TAKEMOTO: Shock Wave and Condensed Matter Research Center, Kumamoto Univ., Japan
Shigeru ITOH: Okinawa National College of Tech., Japan

In recent years in Japan, the demand of cryogenic fluids like a LH2, LNG is increasing because of the advance of fuel cell device technology, hydrogen of engine, and stream of consciousness for environmental agreement. These fuel liquids are cryogenic fluids. Cryogenic fluids have characteristics such as thermal stratification and flashing by pressure release in storage vessel. The mixture of the extreme low temperature fluid and the normal temperature fluid becomes the cause which causes pressure vessel and piping system crush due to explosive boiling and rapid freezing On the other hand, as for fisheries as well, the use of a source of energy that environment load is small has been being a pressing need. And, the need of the ice is high, as before, for keeping freshness of marine products in fisheries. Therefore, we carried out the experiments related to promotion of evaporating cryogenic fluids and generation of ice, in the contact directly of the water and liquid nitrogen. From the results of visualizaiton, phenomena of explosive evapolating and ice forming were observed by using video camera.

Keywords
Cryogenic Fluid, Evaporating, Ice Forming

Irregular diamond-shaped column heating Vortex generated to discussion produced the phenomenon

Corresponding Author
Shuichi Torii
torii@mech.kumamoto-u.ac.jp

Authors
Shuichi Torii, KUMAMOTO And Lin-zijie, KUMAMOTO

The unsteady laminar fluid flow transport phenomena around the diamond-shaped cylinders in free stream are investigated utilizing air flow visualization and numerical methods. Emphasis is placed on the effects of the Reynolds number, Re,and the ratio of cylinder separation distance to length of cylinder ,s/d,on the flow pattern in two cylinders side-by-side arrangement. This present study discloses that the wake characteristics for the cylinders depend on the s/d. The location of diamond-shaped column and temperature the impact on vortex?B

Keywords
Diamond-Shaped Cylinders, Flow Pattern , Gap Spacing ,Laminar Flow , Reynolds Number ,Vortex Shedding

SESSION 2.3

NEW APPLICATIONS

THURSDAY 09 DECEMBER 2010
14:00 – 15:30

CHAIR
T Watanabe
National Fisheries University
Japan

Energy Absorption Capacity of Trailer Under-ride Guard

Corresponding Author
Basem Alzahabi
balzahab@kettering.edu

Authors
Dr. Basem Alzahabi
Department of Mechanical Engineering, Kettering University, USA
Allen Goertz
Department of Mechanical Engineering, Wayne State University, USA

A trailer under-ride guard is usually designed to prevent or minimize the amount of under-ride a passenger vehicle striking the rear of the trailer will sustain. It is typically installed under the rear buck plate of the semi-trailer. The need for an under-ride prevention device is obvious when one realizes that the typical trailer deck is 40 inches above the ground and the car bumper is only 20 inches above the ground.
The most common under-ride trailer guard consists of two vertical members supporting a horizontal member. The vertical members are designed to carry the loading in rear-to-front direction and may be reinforced sometimes by diagonal members. However, most current designs fold completely inward when the under-ride guards are struck with sufficient force. As the guard folds inward, it also pivots upward and eventually disengages from the striking vehicle allowing the vehicle to travel under the trailer, largely unimpeded, until the A-pillars of the passenger compartment are engaged. As such the vehicle occupants are at much greater risk of sustaining severe injuries.
An improved design that is longitudinally offsets the horizontal bumper surface rearward with multiple load bearing supports is being proposed. The support extensions are designed to carry both axial and bending loads. It also allows the bumper to remain lower and engaged with the striking vehicle bumper as the verticals fold inward during the collision. After the vertical members are completely folded, the longitudinal support extensions will function as vertical support members for a second bumper for additional vehicle under-ride.
This paper will present a comparative study between both designs by examining the overall energy absorption capacity of both designs using finite element analysis. The energy absorption will measured from start through ultimate failure. The designs will be modeled with similar strength materials which will be typical of current production bumpers. The finite element models will be created using Hypermesh while the simulations will be performed using LS-Dyna.

Keywords
Energy Absorption Capacity, Trailer Under-ride Guard, Crashworthiness

Prediction of Load-Displacement Curve for Weld-Bonded joints for dissimilar materials and thickness

Corresponding Author
Essam Albahkali
ebahkali@ksu.edu.sa

Authors
Essam A. Al-Bahkali (1)
Jonny Herwan (2)
Department of Mechanical Engineering
College of Engineering
King Saud University
P.O. Box 800, Riyadh 11421, Saudi Arabia
(1) ebahkali@ksu.edu.sa
(2) jonny.herwan@gmail.com

Weld-bonding is a combination of the conventional resistance spot welding and adhesive-bonding. It has gathered wide acceptance as an effective joining process for significant improvement in static, dynamic and impact toughness properties of sheet metal joints. It also improves the corrosion, noise resistance and stiffness of the joint, over those observed in conventional resistance spot welding. In order to get the effective mechanical properties of resistance spot welding, the effect of welding current, welding time, and applied load were studied, the electrode force and the welding current had the significant effect to the maximum tensile shear strength.
Getting a representative model of weld-bonded metals is very important for further design of joining. In this present work, a three-dimensional finite element modeling of dissimilar metals and thickness are developed using ABAQUS package. The process of bonded, spot welding, and weld-bonded of dissimilar materials and thickness joints were studied for a specific adhesive thickness. The ductile fracture limit criteria were developed to predict the deformation and fracture initiation of the model. Detailed material properties of each zone of resistance spot welding (nugget, heat affected zone, and base metal) are essential to accurately simulate the model. The load-displacements curves for different combination models are obtained.

Keywords
Weld-bonded, Load-Displacement curve, Dissimilar Materials, Finite Element Model

Coupled Gas Flow, Diffusion and Reaction in a ppb-level SnO2-NiO Formaldehyde Sensor System

Corresponding Author
Ken Darcovich
ken.darcovich@nrc-cnrc.gc.ca

Authors
K. Darcovich, J.J. Tunney, J.L. Dunford, L.M. Styles, G. Xiao 1 and A. Constant 2
National Research Council of Canada, Institute for Chemical Process and Environmental Technology, Ottawa, Ontario, Canada, K1A 0R6
1 National Research Council of Canada, Institute for Microstructural Sciences, Ottawa, Ontario, Canada, K1A 0R6
2 ICAM-Nantes, 35 av du Champ de Manoeuvres, 44470 Carquefou, France

Semiconducting solid oxides have been long established as materials useful for gas sensing applications. Their functionality arises from electroceramic properties which are highly sensitive to small changes in analyte concentrations. In brief, when a gas to be detected (analyte) at a concentration above some threshold value comes in contact with the solid oxide material, it diffuses into its porous bulk, and simultaneously reacts with the solid oxide surface in a way that serves to appreciably alter the material conductivity. The change in solid oxide electrical conductivity owing to the presence of analyte is the sensor response.
In real sensor applications, a further consideration is the interface between the external gas source and the sensor surface. Models to date which have looked at the internal chemistry and physics of sensor behaviour, have treated the external gas phase as having constant properties. It is known however, that if analyte is delivered to the sensor surface in some kind of flowing system, and temporally, (ie; a desorption pulse from a pre-concentrator), the external transport of the gas and analyte to the sensor surface has a significant influence on the sensor response.
The present project is part of an effort towards the development of a ppb-level formaldehyde sensor system for indoor air quality. An electroceramic response model for a SnO2-NiO composite solid oxide sensing material of mixed p- and n-type conductivity inside the porous ceramic, is coupled with a transient flow field simulation of the delivery of a formaldehyde pulse to the sensor surface. Coupled volumetric CFD domains are used to examine the effects of the transport of formaldehyde from the pre-concentrator to the sensor, and its implications for interpreting the sensor signals from laboratory experiments.

Keywords
Gas sensor, coupled transport and electroceramic properties, ppb-level formaldehyde

Numerical evaluation of pressurizing conditions on explosive welding technique using reflected underwater shock wave

Corresponding Author
Akihisa MORI
makihisa@mec.sojo-u.ac.jp

Authors
Akihisa MORI, Kazumasa SHIRAMOTO and Masahiro FUJITA
Department of Mechanical Engineering, Faculty of Engineering, Sojo University,
4-22-1 Ikeda, Kumamoto 860-0082, Japan

An explosive welding technique using reflected underwater shock wave was developed by some of authors. In this technique, underwater shock wave was generated and propagated when a little high-explosive set on the one side of the specimen was detonated. An underwater shock pressure applied to the specimen was decreasing as an underwater shock wave was propagating from the starting point of detonation. To prevent the attenuation of shock wave, the use of reflector, which is set to taper off for a horizontal direction, was suggested. In this method, it is important that the influence of the geometrical setup of the reflector and the explosive holder is clarified. Therefore, this investigation intends to clarify the pressurizing conditions in changing the inclined angle of the reflector and the explosive holder based on the numerical analysis. Further, a technique of designing a new form assembly is proposed through numerical method.

Keywords
Explosive welding, Reflected underwater shock wave, Numerical analysis, AUTODYN

SESSION 2.4

IMPACTS AND EXPLOSIONS

THURSDAY 09 DECEMBER 2010
16:00 – 17:30

CHAIR
S Itoh
Okinawa National College of Technology
Japan

Research on initiation sensitivity of solid explosive

Corresponding Author
Nobuyuki Matsuo
nobuyuki@shock.smrc.kumamoto-u.ac.jp

Authors
Masahiko Otuka, Asahikasei Chemicals Corporation
Hideki Hamasima, Kumamoto Industrial Research Institute
Kazuyuki Hokamoto, Shock Wave and Condensed Matter Research Center, Kumamoto University
Sigeru Itoh, Okinawa National College of Technology

In the processing using high energy of explosive, there are various application examples including explosive welding, dissimilar metal plates and powder compaction of diamond. In these examples mainly high sensitive explosive was used. High sensitive explosive is dangerous and accidental explosion is possible to happen suddenly. Recently, the explosive development was more focused on high safety than high power. Also, when we produced the explosive, it is necessary to manufacture with low cost and high safety. It is important to figure out explosive characteristics for utilizing explosive safely and efficiently.

In this research, we focused on initiation sensitivity of solid explosive in explosive characteristics and we performed numerical analysis of sympathetic detonation. Sympathetic detonation experiment are difficult for clean up because the explosive might not detonate. So, we chose numerical analysis. Numerical analysis was conducted to narrow down experimental conditions for performing fewer experiment and avoid high risk. So we tried to do numerical analysis for evaluating sympathetic detonation. We performed numerical analysis using LS-DYNA for impact shock matter analysis can be solemnized. As a reaction equation, Lee-Tarver model was elected, the impact detonation process was numerical value analyzed by the ALE code. Analysis model is quarter circular cylinder part. From above SEP as a donor explosive, under that aluminium, steel, air, SEP as an acceptor explosive were set by rotation. The shock wave was generated by donor explosive attenuates while passing PMMA layer, air layer, metallic layer and so on. Then we evaluated if initiation of acceptor explosive occurred when it enters into the acceptor explosive. This is evaluated from the attenuation of impact pressure, shock wave incident pressure to acceptor explosive, reactive rate of acceptor explosive, and detonation pressure.

Keywords
Initiation sensitivity, sympathetic detonation, Lee-Tarver model, shock wave

Impact of explosive to civil and underground structures

Corresponding Author
Dolezel, V.
vladimir.dolezel@upce.cz

Author
Dolezel, V.,
Universita Pardubice, Studenstska 95, 53210 Pardubice, Czech Republic
Prochazka, P.,
Association of Czech Civil Engineers, Prague, Czech Republic

The submitted paper will be concerned with dynamical problems prevailingly in assessment of underground structures and structural elements, which are loaded by explosion. It appears that the material obeys time-dependent loading, which is decisive in design of protective structures. This is mainly done by terrorist attacks which are considered nowadays very often. The aim of the project is to involve relatively new numerical methods into computation, which work with physical phenomena. In materials due to change of a concentrated force loading (Sedov) or description using shock waves can be preferred. An attack against underground railway and station is of great interest to designers of such structures.
In numerical modeling we suppose to assess the structures by FEM and continue developing certain discrete element methods (free hexagons, for example), meshless methods (boundary elements, smooth hydrodynamics method, manifold method, etc.). Experimental part is provided by scale models, which are available in Transport Faculty in Pardubice. Moreover, additional experiments will be presumed to be conducted in cooperation with certain external organizations (Nanyang Technological University Singapore).
The objective is to study sudden time dependent changes in the materials of underground structure due to impacts against structures or shocks caused by a sudden release of accumulated energy in a mass. The formulations using Hamilton?s principle instead of Lagrange?s principle is prepared. Influence of impact load against underground structures, i.e. bridges, hangars, envelopes of underground nuclear power stations, for example, is described. The experiments are conducted in scale model 1:10. The mechanism of transformation of the structures will be recorded by digital high speed camera. It obviously appears hat the higher fiber volume ration the better protection against the impact load. This is quantifiably evaluated in the presentation. The theory presented in this paper stems from relatively large range of tests.
We would like to acknowledge financial support of GACR, project number 103/08/0922 and 103/08/1197. The second author was also sponsored by CZE MSM 6840770001.

Keywords
Dynamical problem, underground structures, terrorist attacks, impact load

Influence of fire in underground structures

Corresponding author
Dolezel, V.
vladimir.dolezel@upce.cz

Author
Dolezel, V.,
Universita Pardubice, Studenstska 95, 53210 Pardubice, Czech Republic
Prochazka, P.
Association of Czech Civil Engineers, Prague, Czech Republic

Fiber reinforced concrete (FRC) exposed to extremely elevated temperature is of great concern in tunnel structures, for example. In the paper a study on extreme damage on the face of FRC block being subjected to fire of 12000C in duration of 2 hours is to be presented from both theoretical and experimental views. The experiments have been conducted in a furnace its dimensions are to be specified in the paper.
The combination of concrete and reinforcing fibers is generally a very gentle problem. It is now commonly recognized that the actual behavior of concrete subjected to high temperature is a result of many factors, including both environmental factors and constituent materials. Among the mechanical factors, the heating rate and peak temperature are the two main factors which have a significant influence on the concrete properties. This is particularly valid in the applications to fire defense assessment, as on one hand side a natural requirement is a high strength of the entire material and on the other hand side the resistance against boosting evaporation at extremely elevated temperature in the pores of the material has to be increased.
Material properties can be defined by different agents. Under fire conditions, materials are subjected to transient processes and therefore there is a big need to quantify these properties being determined under transient conditions. These properties should be distinguished from other properties derived under steady state conditions. Here basalt fibers are used with fiber volume ratio of 1%.
During the heating of a concrete face the tensile strength cannot be exceeded and if so, during the cooling process the tensile strength descends to zero and the admissible set of displacements turns to a cone. The volume changes, pore pressure, effect of overheated vapor, all these phenomena can be involved in eigenparameters. They appear in the generalized Hookes law as an additional term to strains (eigenstrain) or to stresses (eigenstress). We acknowledge financial support of GaCR project P104/10/1021 and P103/08/1197.. The second author was also supported by the grant project CZE MSM 6840770001.

Keywords
Fiber reinforced concrete, temperature, pore pressure, eigenparameters

Fluid-Structure Interaction Analysis of a Steel Pipe Subjected to Internal Blast Loading

Corresponding Author
Tei Saburi
t.saburi@aist.go.jp

Authors
Tei Saburi, Shiro Kubota and Yuji Ogata
National Institute of AIST
Onogawa 16-1, Tsukuba, Ibaraki 305-8569, Japan

This paper presents numerical fluid-structure interaction analysis of dynamic behavior of a steel pipe under internal blast loading. We have been studying the influence of blast loading onto structures in order to evaluate explosion safety. It helps to develop blast protection and mitigation techniques and design protective structures such as blast wall and explosives storage devices. We have been conducted experimental study as well as developed CFD code to simulate and understand explosion phenomena. A Semi-Lagrangian CFD code of MARS has capability of simulating initiation and detonation of explosives, blast propagation, and so on, but it is not functional to analyze dynamic structural behavior. In order to acquire the capability of the fluid-structure interaction analysis, a implementation of CSD (computational structure dynamics) simulating function was considered. An explicit Finite-Element structure solver DYNA3D 2000, which has been applied to transient dynamic problems as public code, is suitable to couple structure solving capability with MARS, so a FSI coupling code for MARS fluid solver and DYNA3D 2000 structure solver was developed. The developed FSI code was adapted to the problem of dynamic response of the steel pipe under internal blast loading. Air and explosive was modeled using ideal gas EOS and the structure was modeled using Johnson-Cook strength model and Mie-Gruneisen EOS. Explosive has density of 1400kg/m3 and energy of 5.29J/kg, and other numerical parameters for calculation were used typical value. Strain change and radial displacement of circumference were evaluated, and FSI code was validated comparing the numerical results with experimental results.

Keywords
Fluid-structure interaction, explosion, blast loading, finite element method, CIP

SESSION 3.1

THERMAL ENGERY 1

FRIDAY 10 DECEMBER 2010
09:30 – 11:00

CHAIR
S Torii
Kumamoto University
Japan

Investigation of Size Effects to the Mixing Performance on the X-shaped Micro-Channels

Corresponding Author
Shu-Min Tu, Kumamoto University

Due to the developing of micro-electro-mechanical-system, MEMS, the fabrication of the microminiaturization devices becomes obviously important. The advances in the basic understanding of fluid physics have opened an era of application of fluid dynamics systems using micro-channels. The purpose of this study is to research the flow transport phenomenon by employing different kinds of micro-channel sizing in X-shaped micro-channels. As the working fluid, water is injected to micro-channel at different mass flow rate. Over a wide range of flow condition, $1.06 < Re < 514$, in X-shaped micro-channels, the mixture performances of numerical simulation, flow visualization, and temperature distribution remain the same. At the same mass flow rate as the Reynolds number below 112.53, the biggest channel size had the slowest flow velocity and got the best mixing performance; as the Reynolds number above 112.53, the smaller the channel sizing, the lower the pressure drops and the faster velocity becomes. The transition form early from laminar flow, the unsteady flow is an advantage for mixing in the limited mixing area, therefore 0.7 mm got the best mixing performance. It is clear that the size of the channel plays an important role in the X-shaped micro-channels.

Thermal Fluid Flow Transport Characteristics in confirmed channels with two-dimensional duel jet impringement

Corresponding Author
C. Senkal, Kumamoto University

The flow and heat transfer characteristics of laminar dual circular jet impinging on a heating plate with inclined confinement surface has been investigated numerically.Governing equations in steady state are solved by a control volume based finite-difference method. The simulations have been carried out for reynolds number (250≤Re≤1000), the angle of inclination of the confined upper wall (0o ≤ θ ≤ 10o), circular jet to annular jet velocity ratio (0≤VR≤2) and jet to target plate distances between 2D and 8D where D is the outer diameter of dual jet.SIMPLE algorithm was used to obtain velocity and temperature fields. Hybrid difference scheme is adopted for the discretized terms in the governing equations. The discretised equations are solved iteratively using the tridiagonal matrix algorithm line solver. Heat transfer performance along the heated wall is amplified with an increase in the velocity ratio and the reynolds number.On the contrary,a substantial reduction in the heat transfer rate, for VR=0.0, occurs in the stagnation zone, because the absence of the inner nozzle injection causes the recirculation in the corresponding region. The heat transfer rate in the stagnation zone is attenuated by increasing the jet nozzle to impinging plate distance. In particular, the effect of the inclination angle in the downstream region,especially at the vicinity of outlet, is major then other effects Nusselt number distribution on the impingement plate is affected by inclined upper wall because inclination of the wall accelerates the exhaust flow. The streamwise reduction in the heat transfer rate for θ=0.0o is suppressed by the presence of the inclined confinement surface and its value is intensified by the inclination angle.

Development of New Model Combustor for Biomass

Corresponding Author
Masato Urashima, Kumamoto University

In general, biomass is a renewable energy source because the energy that it contains comes from the sun. One of sources of biomass is municipal solid waste. The final goal of the study is to develop the combustor for the micro gas-turbine using the biomass as a fuel. Here, it is very important to remove ashes (10μm or more in diameter) in the gas because its size affects the strength of the turbine blade. The aim of the present study is to observe the combustion phenomena relevant to a mixture of waste fluid and waste oil. Emphasis is placed on the ash size which is produced from the combustion chamber developed here.
The experimental apparatus consists of the combustion chamber which is the shape of the cyclonic separator and the fuel injection device. Fresh ambient air, as a result of air entrainment subsequently triggered by the formation of ejected jet flames, is brought into the jet flow, which mixes with reactants and the hot products of chemical reaction that proceeds in the flames. The ash exhausted from the outlet of the combustion chamber is captured by the paper seat and it is observed with the microscope. The exhaust gas inclued large size ashes which far exceeds 10 mm but almost ash's size is smaller than 10 mm
An experimental study has been performed on the combustion phenomena relevant to a mixture of waste liquid and waste oil.Emphasis is placed on the ash size which is produced from the combustion chamber developed here.
In general, biomass is a renewable energy source because the energy that it contains comes from the sun. One of sources of biomass is municipal solid waste. The final goal of the study is to develop the combustor for the micro gas-turbine using the biomass as a fuel. Here, it is very important to remove ashes (10μm or more in diameter) in the gas because its size affects the strength of the turbine blade. The aim of the present study is to observe the combustion phenomena relevant to a mixture of waste fluid and waste oil. Emphasis is placed on the ash size which is produced from the combustion chamber developed here.
The experimental apparatus consists of the combustion chamber which is the shape of the cyclonic separator and the fuel injection device. Fresh ambient air, as a result of air entrainment subsequently triggered by the formation of ejected jet flames, is brought into the jet flow, which mixes with reactants and the hot products of chemical reaction that proceeds in the flames. The ash exhausted from the outlet of the combustion chamber is captured by the paper seat and it is observed with the microscope. The exhaust gas inclued large size ashes which far exceeds 10 mm but almost ash's size is smaller than 10 mm
An experimental study has been performed on the combustion phenomena relevant to a mixture of waste liquid and waste oil.Emphasis is placed on the ash size which is produced from the combustion chamber developed here.

A Basic Research of Separation and Collecting Solid Materials from Solid-Liquid

Corresponding Author
Koichi Shimamoto, Kumamoto University

An experimental study is performed on the collection of rare metals with the aid of the cyclone separator. Emphasis is placed on optimum condition such as heating time, heating rate, solid-liquid flow for high collection rate pertinent to complete separation between water and glass beads from its mixture. It is observed that sizes of solid and liquid particles which are injected to the medium should be small. In order to get a spray flow, surface area of liquid and solid particle are set to large and particles gain heat much more with this method. Thus water evaporates and only dried glass beads fall the bottom of cyclone separator. The aim of this study is collecting dried rare metals from mixture of particles of rare metals and coolant. The methods of collecting rare metals have taking electronic devices to pieces, and attaching particles of rare metals to marine alga and so on. It is focused that when machine components are manufactured, it is generated that small particles such as grinding dust with coolant. These waste fluids are usually condemned. Some machine components include rare metals. It is subjected that collecting rare metals from these waste fluids, and recycle the collected rare metals. In this study, in order to collect rare metals with the cyclone separator, it is examined to change heating time, heating rate, solid-liquid flow and found out the high collection rate. And in this research, liquid is substituted to water, particles are substituted to glass beads. And following experiments are carried out; finding the optimum conditions for high collection rate and complete separation of this water and glass beads.

SESSION 3.2

THERMAL ENERGY 2

FRIDAY 10 DECEMBER 2010
11:30 - 13:00

CHAIR
S Torii
Kumamoto University
Japan

Response Examination to Make Micro-pump with the Aid of Peltier Devices and Heat Deformation Material

Corresponding Author
Yasuhito Takakura, Kumamoto University

The micro-pump is one of the application research and development fields of the MEMS technology. Its fundamental form is proposed from the end of the 1980's to the beginning of the 1990's with the MEMS technology development. The aim is to design and develop a cylindrical valveless pump, which is driven by expansion and contraction of the heat deformation material with the aid of heating and cooling properties of peltier devices. And we want to make micro-pump of this type. The present pump's size is 20mm x 43mm (diameter x length) and it consists of the diffuser valve unit, the heat deformation material unit, the nozzle valve unit, the peltier devices, and the cover. The input current of the peltier devices is controlled by the bipolar power supply so that both faces of the peltier devices are heated and cooled periodically. Correspondingly the internal diameter of the heat deformation material unit is changed because of heat expansion and contraction of its material. So pump can operate combined with shapes of both the diffuser valve and the nozzle valve. The working fluid filed in the micro-pump flows by the periodical material deformation. In the present study, internal space of the heat deformation material unit is recorded by X-ray apparatus and the fluid flow in the micro-pump is measured by PIV. Working fluid is water and particles for PIV are air bubble. As a result, we can see that working fluid and bubbles repeats going on and back in the pump. And it moves relatively in the pump.

Transport phenomenon Vortex behind irregular diamond-shaped column heated under constant heat flux

Corresponding Author
Lin-zijie, Kumamoto University

The unsteady laminar fluid flow transport phenomena around the diamond-shaped cylinders in free stream are investigated utilizing air flow visualization and numerical methods. Emphasis is placed on the effects of the Reynolds number, Re, and the ratio of cylinder separation distance to length of cylinder, s/d, on the flow pattern in two cylinders side-by-side arrangement. The present study discloses that the wake characteristics for the cylinders depend on the s/d.

Effect of Plate Shape on Thermal-Fluid Flow Characteristics in Plate Heat Exchanger

Corresponding Author
Keita Izumi, Kumamoto University

In general, the plate heat exchanger is comparatively efficient in various heat exchangers and it is widely used for many industrial applications. The plate heat exchanger consists of the thin plate, the frame, and the gasket in which the plate shape is a key factor the heat transfer enhancement. It is important to develop the plate which suppresses the frictional pressure drop and at the same time promotes the heat transfer rate. The aim of the present study is to study thermal-fluid flow characteristics in model of plate heat exchanger with various plate shapes. Consideration is given to enhancement of heat transfer and attenuation of pres-sure drop in the single plate model of the plate heat exchanger. The results of various plates are compared with them of flat and herringbone plates and the optimum shape of the heat transfer plate in model of plate heat exchanger is reported. Experimental apparatus consists of the modeled plate heat exchanger, in which thin aluminum or titanium plate is sandwiched by two large side walls made of acrylic windows for illumi-nation and observation. Water is used as the working fluid in the vertical heat exchanger channel. A down-ward stream of cold water flows between the vertical plate and the acrylic window, while hot water flows upward along the other side wall of the vertical plate. The fluid temperature and pressure drop are meas-ured at the test section inlet and outlet of each channel with the aid of thermocouples and differential pres-sure gauges, respectively. The overall heat transfer coefficient and frictional pressure drop were obtained by measuring temperature and pressure drop between inlet and outlet of the channel, respectively.

New simulation technique of the LSC Jet

Corresponding Author
Hitoshi Miyoshi
miyoshi@impact-lab.jp

Author
Hitoshi Miyoshi
Impact Engineering Laboratory

Calculation region of multi material Euler solver on the AUTODYN-3D was effectively expanded beyond the initial boundaries. Although linear shaped charge (LSC) jet is elongated toward the target on the simulation, the upper part of the Euler region becomes needless and rejectable after the calculation proceeds. The new technique we developed provides effectiveness of the Euler region and the continuing calculation method, corresponding to elongation of the LSC jet.

Keywords
Linear Shaped Charge Jet, Euler Region Expanding, High Speed Video Image

SESSION 3.3

FOOD PROCESSING

FRIDAY 10 DECEMBER 2010
14:00 – 15:30

CHAIR
K Hokamoto
Kumamoto University
Japan

The design of rice powder production vessel and the pulverization of the rice which used numerical simulation

Corresponding Author
Manabu Shibuta
manabu@shock.smrc.kumamoto-u.ac.jp

Authors
Manabu Shibuta(1), Hideki Hamashima(2), Shigeru Itoh(3)
(1) Graduate School of Science and Technology, Kumamoto University, Japan
(2) Kumamoto Industrial Research Institute, Japan
(3) Okinawa National College of Technology, Japan

In recent years, the food self-support rate of Japan is 40%, and this value is the lowest level of major advanced country. The stable supply of food is a big subject that Japan has. Therefore, rice powder attracts attention for improvement of the food self-support rate in Japan.
Previously, the rice powder is produced by two methods. One is dry type, and the other is wet type. However, these systems have a fault of the heat damage of the starch and the consumption of a large quantity of water. In our laboratory, as solution of those problems, production of the rice powder by using the underwater shock wave is considered.
Shock wave is the pressure wave which is over velocity of sound by discharging high energy in short time. Propagating shock wave in water is the underwater shock wave. This food processing using an underwater shock wave has little influence of heat and its processing time is very short, preventing the loss of nutrients.
In this research optical observation experiment and the numerical simulation were performed using trial vessel, in order to understand the behavior of the underwater shock wave in the development of the rice powder production vessel using an underwater shock wave at the factory. In addition, in order to understand the rice powder production and to develop it, the numerical simulation about pulverization of rice is performed. By this method, the pressure which takes for rice at the time of pulverization, and its pulverization phenomenon are solved. Analysis soft LS-DYNA was used for these numerical simulations.
The comparative study of the experiment and the numerical simulation was investigated. The behavior of the shock wave in the device and transformation of rice were able to be clarified.

Keywords
Underwater shock wave, Numerical simulation, LS-DYNA, Food processing, pressure vessel, rice powder

The Basic Research for Pulverization of Rice Using Underwater Shock Wave by Electric Discharge

Corresponding Author
Makoto Ide
ide@shock.smrc.kumamoto-u.ac.jp

Authors
Makoto Ide, Graduate School of Science and Technology, Kumamoto University, Japan
Manabu Shibuta, Graduate School of Science and Technology, Kumamoto University, Japan
Naoyuki Wada, Graduate School of Science and Technology, Kumamoto University, Japan
Shigeru Tanaka, Faculty of Engineering, Kumamoto University, Japan
Hideki Hamashima, Kumamoto Industrial Research Institute, Japan
Shigeru Itoh, Okinawa National College of Technology, Japan

In recent years, the food self-support rate of Japan is 40%, and this value is the lowest level of these countries. Therefore, rice powder attracts attention for improvement of the food self-support rate in Japan.
Previously, the rice powder is manufactured by two methods. One is dry type, and the other is wet type. The former is the method getting rice powder by running dried rice to rotating metal, and has a problem which that starch is damaged by heat when processing was performed. The latter is performed same method against wet rice, and has a problem which a large quantity of water is used. As a method to solve these problems, an underwater shock wave is used.
Shock wave is the pressure wave which is over speed of sound by discharging high energy in short time. Propagating shock wave in water is underwater shock wave. The characters of underwater shock wave are long duration of shock wave because water density is uniform, water is low price and easy to get and not heat processing.
In this laboratory, the research about pulverization of rice using underwater shock wave by electric discharge is performed. As the result, a good quality powder with a little starch damage can be obtained. However, because getting the smaller grain rice powder is very inefficient, it is impossible to mass-produce it.
Then, it is purpose of this research to crush rice efficiently by suppressing the basis of the phenomenon of the underwater shock wave by optical observation about propagating of underwater shock wave by electric discharge and pressure measurement, and applying the data to the numerical analysis.
As the experiment result, we could confirm that the underwater shock wave propagated.

Keywords
Rice powder, self-support rate, underwater shock wave, electric discharge, optical observation, pressure measurement, numerical analysis

On the particle size distribution and the pressure condition of the rice powder manufactured from the instantaneous high pressure

Corresponding Author
Takemoto, Ayumi
tkmt@mech.kumamoto-u.ac.jp

Authors
Takemoto, Ayumi*, Kudou, Yasufumi**, Mimaki, Nami**, Shigeru, Itoh***
* Shock Wave and Condensed Matter Research Center, Kumamoto University
** Kumamoto Industrial Research Institute
*** Okinawa National College of Technology

The instantaneous high pressure, that is, the shock wave divides into the penetration wave and the reflected wave on the surface of the density difference.The shock wave divides into the penetration wave and the reflected wave at the surface of the density difference. The reflected wave caused on a high density side brings the pull stress from negative pressure. And, the high-speed destruction phenomenon that is called spalling destruction is caused. The shock wave exceeds the sound velocity, so it makes the instantaneous high pressure within several micro second. Moreover, frictional heat is not caused because it is an instantaneous phenomenon.
As for the rice powder manufactured by using this high-speed destruction phenomenon, the peak of the particle size distribution is sharper than that of existed milling technologies.[1] Relations between the grain size distribution and the pressure value, the processing frequency on the rice powder manufactured by using the instantaneous high pressure are reported by this research.
[1] Takemoto, A., Kudoh. Y., Mimaki, N., and Itoh, S. gOn the processing characteristic of the rice powder by the instantaneous high pressure processing The Annual Meeting of the Japan Society of Cookery Science, 2010, pp.25., 2010

Keywords
Shock wave, spalling destruction, milling

Evaluation using numerical simulation of bubble pulsation in food processing using underwater shock wave

Corresponding Author
Hideki Hamashima
hamashima@kmt-iri-go.jp

Authors
Hideki Hamashima*, Naoyuki Wada**, Manabu Shibuta**, Kazuyuki Hokamoto** and Shigeru Itoh***
*Kumamoto Industrial Research Institute
**Shock Wave and Condensed Matter Research Center, Kumamoto University
***Okinawa National College of Technology

The food processing technology using a shock wave can prevent deterioration of the food by heat because it can process food in a short time. The shock wave used for the food processing has the shock wave which occurs just after the explosion and the shock wave which occurs by the repetition of the expansion contraction of the explosion product gas (bubble pulsation) after an explosion. In order to process safely, it is important to clarify the behavior of the shock wave because the action to the food by these shock waves is complicated. On the other hand, since the numerical simulation of a 3D-model can be easily performed by remarkable development of computer technology in recent years, a designer is utilizing analysis technology (CAE; Computer Aided Engineering) as frontloading. However, since the numerical simulation about a shock wave is not performed widely, it is important to evaluate the validity of the analysis method.

In this research, in order to investigate the behavior of the bubble pulsation generating the shock wave in the container used for food processing, the numerical simulation and the optical observation experiment were performed. The numerical simulation about the bubble pulsation generated by explosion was performed using analysis software LS-DYNA. In the experiment, the bubble pulsation was observed with the high-speed video camera.

Comparing and examining were performed about the numerical simulation result and the experimental result. In the numerical result, it was demonstrated that the behavior of bubble pulsation changed by the reason that the restraint of the flow of water changed by the difference of the boundary condition of the circumference surface of a calculation region. The numerical result and the experimental result were well in agreement, and the knowledge about the validity of the numerical simulation in this research was obtained.

Keywords
Underwater shock wave, bubble pulse, Food processing

www.aau.ac.uk

www.**kumamoto-u.ac.jp**

www.jaguar.com

www.univ-lille1.fr/gdr-ifs

www.hin.no

www.hsl.gov.uk

www.physica.co.uk

www.salford.ac.uk

www.multi-science.co.uk

www.multiphysics.org