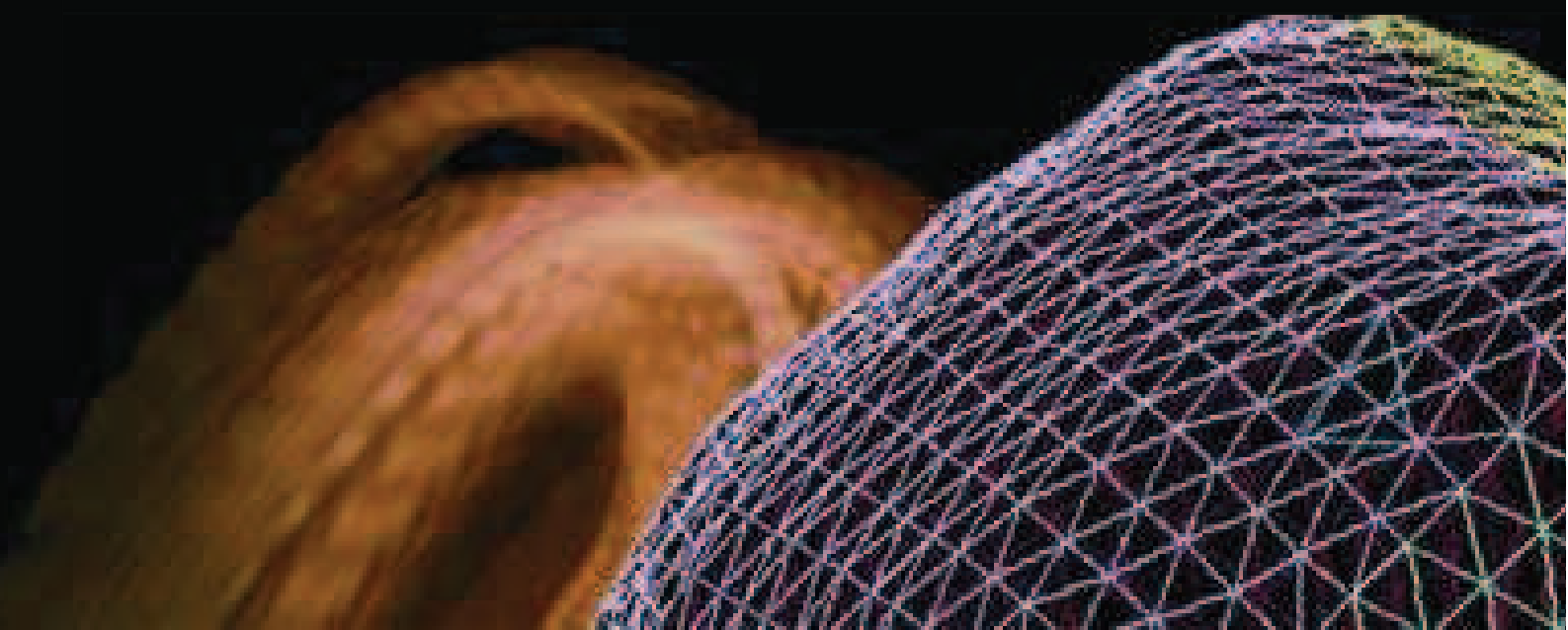
ISSN (Online) 2409-1669
ISSN (Print) 2409-7527

MULTIPHYSICS 2007

12 - 14 December 2007
Manchester, UK



MULTIPHYSICS 2007
12-14 December 2007
Manchester, UK

Conference Board

Chair

Moatamedi M, University of Salford, UK

Vice-chair

Ewan BCR, University of Sheffield, UK

Scientific Committee

Chynoweth S, *Shell Global Solutions, UK*
Cross M, University of Wales, Swansea, UK
Elger G, Electrolux Major Appliances Europe, Italy
Erchiqu F, University of Quebec, Canada
Gabrys J, Boeing, USA
Holdo A, University of Hertfordshire, UK
Itoh S, Kumamoto University, Japan
Kilic S, Bogazici University, Turkey
Moodie K, Health & Safety Executive, 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
Sigrist J, DCN Propulsion, France
Souli M, University of Lille, France
Tabiei A, University of Cincinnati, USA
Tehrani A, Serco Assurance, UK
Wakes S, University of Otago, New Zealand
Zeguer T, Jaguar, UK

General Information

Scope of Conference

Multiphyiscs analysis has become increasingly important in recent years due to demand in design and understanding of complex systems.

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, 12th December	13:00-18:00
Thursday, 13th December	08:00-17:00
Friday, 14th December	08:00-17:00

Special Events

- Wednesday 18:00
  Welcome Reception
- Thursday 19:00
  Conference Banquette

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

The Conference Board would like to thank the sponsors for their support.

Keynote Speaker

Professor WB Zimmerman
University of Sheffield, UK

BIOGRAPHY

William Zimmerman, Professor of Biochemical Dynamical Systems, is a graduate of Princeton and Stanford Universities in Chemical Engineering, the author of over 80 scientific publications, past Director of the MSc in Environmental and Energy Engineering, and a winner in US and UK national competitions of five prestigious fellowships (2005-6 Royal Academy of Engineering Senior Research Fellow, 2000-5 EPSRC Advanced Research Fellow; 1994-99 Royal Academy of Engineering, Zeneca Young Academic Fellow; 1991-93 visiting professorships appointed by the French Ministry of Education at the Littoral University in Dunkirk in 1998 and 2005. He has been principal or co-investigator in research projects worth over £3.1m since 1994 and supervised over 40 researchers directly. His work relevant to this programme area is in the design of microfluidic reactors and separators with finite element methods. EPSRC *Newsline* recently did a review article on his fellowship programme. His Royal Academy of Engineering SRF, entitled "Systems Biology and Chemical Engineering" has laid down a protocol for optimal experimentation for bioreactor model-building via inverse methods. He directed Microfluidics 2003, a workshop at CISM in Udine from which he edited a book on Microfluidics, and is the author of two bestselling texts on multiphysics modelling. He recently spoke about his programme in microfluidics as a plenary lecturer at an industry forum organised by TUV-NEL in Glasglow. He has patented (UK0621561) a microbubble generator in 2006-7, and is currently applying for another on microfluidic plasma reactors.

MULTIPHYSICS 2007

Programme

Time	Wednesday 12 Dec 07	Thursday 13 Dec 07	Friday 14 Dec 07
09:00 – 10:30		Session 2.1 *Impact Dynamics*	Session 3.1 *Software Developments*
10:30 – 11:00		Coffee Break	
11:00 – 12:30		Session 2.2 *Shock and Explosives*	Session 3.2 *Heat Transfer and Thermodynamics*
12:30 – 13:30		Lunch	
13:30 – 15:00	Registration	Session 2.3 *Micromechanics*	Session 3.3 *Fluid Structure Interaction*
	Keynote Address		
15:00 – 15:30	Tea Break		
15:30 – 17:00	Session 1.4 *Flow Modelling and Simulations*	Session 2.4 *Electrophysics*	Session 3.4 *Modelling Advances*
18:00	Welcome Reception		
19:00		Conference Banquette	

Full Programme

Wednesday 12 December 2007

13:00 **Registration**

14:00-15:00 **Keynote Address**

Chair: M Moatamedi, University of Salford, UK

Prof WB Zimmerman,
University of Sheffield, UK

Optimal Multiphysics Modelling and Experimentation in Microfluidic Chemical Technology Design.

WB Zimmerman and JM Rees
University of Sheffield, Sheffield, UK

15:00-15:30 **Tea Break**

Wednesday 12 December 2007

15:30-17:00 Session 1.4 *Flow Modelling and Simulations*

Chair: BCR Ewan, University of Sheffield, UK

Numerical Analysis of Fluid mixing in T-Shape Micro Mixer

M.S. Virk , A.E. Hold

University of Hertfordshire, UK

Numerical Simulation of Boundary Heat Flux during Alloy Solidifcation and its Effect on Natural Convection, Macrosegregation and Microstructure evolution.

K.V. Sreenivas Rao , G. Phankumar, T.S. Prasanna Kumar

Indian Institute of Technology Madras, India

Study of Air-Flow and Heat Transfer on a Large-Scale Roller-Kiln via Computational Simulation and Experimental Prototype

A.Chai, A.Vakhguelt

Swinburne University of Technology, Malaysia

Understanding in Multiphysics of the Unsaturated Flow in Porous Media

Y. Wang, M. Moatamedi

University of Salford, UK

18:00 Welcome Reception

Thursday 13 December 2007

09:00-10:30 Session 2.1 *Impact Dynamics*

Chair: S Chynoweth, *Shell Global Solutions, UK*

Influence of Projectile-Barrell Interactions on Launch Dynamics
M.M. Chen
Army Research Laboratory, USA

Pressure Vessel made by Free Forming using Underwater Explosion
H. Iyama, H. Maehra, Y. Hidaka, S. Itoh, Yatsushiro
National College of Technology and Kumamoto University, Japan

Deformation of Egg-Box Panel subjected to Compressive Load
S. Nowpada , E. C. Chirwa , P. Myler
The University of Bolton, UK

Buckling Failure of Thin Walled Cylindrical Shells Under Axial Compression
H. Ullah
NESCOM, Pakistan

10:30-11:00 Coffee Break

Thursday 13 December 2007

11:00-12:30 Session 2.2 *Shock and Explosives*

Chair: Z Ren, University of Maribor, Slovenia

The Numerical Analysis of Food Processing using Underwater Shock Wave

N. Okamoto , A. Oda, S. Itoh, M. Moatamedi
Kumamoto University and University of Salford, Japan and UK

On the Destruction of the Cell Wall of Plants and its Mechanism by the Shock Wave

A. Takemoto, K. Kuroda, H. Iya*ma , S. Itoh*
Yatsushiro National College of Technology and Kumamoto University, Japan

Basic Study on Promotion of Thawing Frozen Soil by Shock Loading

T. Watanabe, H. Maehara, S. Itoh
National Fisheries University and Kumamoto University, Japan

Numerical Analysis of Detonation Phenomena using PBX High Explosives

K. Kuroda ,M. Otsuka , E. Hida, M. Moatamedi, S. Itoh
Kumamoto University and Asahi Kasei Chemicals Co. and University of Salford, Japan and UK

12:30-13:30 Lunch

Thursday 13 December 2007

13:30-15:00 Session 2.3 *Micromechanics*

Chair: S Itoh, Kumamoto University, Japan

Exact Code Scaling

M. Antal, Makai

BME Institute of Nuclear Techniques, Hungary

The Over-Barrier Resonant states and Multi-Channel Scattering by a Quantum Well

A.F.Polupanov, V.I.Galiev, A.N. Kruglov

Institute of Radio-Engineering & Electronics of the Russian Academy of Sciences, Russia

Study on Low Velocity Detonation Phenomena in Nitromethane

A. Osada, H. Hamashima, Y. Kato, S. Itoh

Kumamoto University and AIST and Nippon Koki Co Ltd, Japan

Fission Models and Odd Even Effect in Charge in Low-Energy Fission

G. Medkour Ishak Boushaki, M. Asghar, N. Boucheneb, P. Geltenbort, G. Barreau , M. Allab

Institute of Physics, Algeria

15:00-15:30 Tea Break

Thursday 13 December 2007

15:30-17:00 Session 2.4 *Electrophysics*

Chair: AE Holdo, Coventry University, UK

Multiphysics Simulation of a Micromirror Device
S. Kini, K. Shah, M. Megahed
ESI Group, Germany

In-Orbit Attitude Actuation Using Solar Panels
R. Varatharajoo, T.M. San
University Putra, Malaysia

Circulation of Slag and Molten Metal in a Submerged Arc Furnace
I. Mc Dougall, J.-H. Grobler
CSIR, South Africa

Gamma Radiation Effect on Enhancement VMOSFET
S. A. Hayat, M.I. Sabir, R.Raza
COMSATS Institute of Information Technology, Pakistan

19:00 Conference Banquette

Friday 14th December 2007

09:00-10:30 Session 3.1 *Software Developments*

Chair: T Morris, NAFEMS, UK

NAFEMS: Engineering Analysis and Simulation
T. Morris
NAFEMS, UK

Multiphysics Modelling and the COMSOL Methodology
K. Howard
Comsol, UK

ANSYS Multiphysics - An Integrated and Unified Virtual Prototyping Tool Kit for Industry and Research
B. Miller
Wilde FEA Ltd, UK

PHYSICA: PHYSICA Approach to Challenging Multiphysics Problems
L. Marks
Physica, UK

10:30-11:00 Coffee Break

Friday 14 December 2007

11:00-12:30 Session 3.2 *Heat Transfer and Thermodynamics*

Chair: A Tehrani, Serco, UK

Virtual Cooking
C. Catalogne, J-Y. Noel, L. Padovan, G. Elger
Electrolux Major Appliances Europe, Italy

Numerical Prediction of Indoor Temperature Stratification
K. Chow, A. E. Holdø
University of Hertfordshire, UK

A Numerical Investigation on Fluid-Dynamical and Thermo-Chemical Operative Conditions Concurring in Metal Deposition on Post-Combustion Burners
G. Cammarata, S. Caggia, L. Cammarata, G. Petrone
University of Catania, Italy

Modeling the Thermal Behavior of Fluid Flow inside Channels Using an Artificial Locally Linear Neuro-Fuzzy Approach
A. Hashemian, T. Babaie, C. Lucas
Sharif University of Technology, Iran

12:30-13:30 Lunch

Friday 14 December 2007

13.30-15:00 Session 3.3 *Fluid Structure Interaction*

Chair: M Souli, University of Lille, France

Computational Simulations of a Spacecraft Water Landing and the Water Impact of a Sphere

M. Vesenjak, Z. Ren, M. Moatamedi
University of Maribor and University of Salford, Slovenia and UK

On the Coupled Dynamics of Wave-Seabed Interaction

S. Williams
The University of Sydney, Australia

Explosive Forming of Stainless Steel Plate by Under Water Shock Wave

Y. Hidaka, H. Maehara, S. Itoh
Kumamoto University, Japan

Particle Filtration Processes in Deformable Media

M. Matalan, W. Brandstätter, G. Boiger, R. Bouwman
ICE-Strömungsforschung GmbH, Austria

15:00-15:30 Tea Break

Friday 14 December 2007

15.30-17:00 Session 3.4 *Modelling Advances*

Chair: T Rahulan, University of Salford, UK

Modelling of Complex Process Conditions for the Calculation of Welding Residual Stresses and Distortion

M. Urner, E. Hanssen, T. Welters, K. Dilger
Technical University of Braunschweig, Germany

Modeling and Control of a Wheelchair for Mobility on Two Wheels

S. Ahmad
University of Sheffield, UK

The Immersed Boundary-Lattice Boltzmann Method for the Unsteady Fluid-Particle Interaction

S. Farnoush, M.T. Manzari, M.S. Seif
Sharif University of Technology, Tehran, Iran

Fault Diagnosis of Rotating Machineries based on Gabor Order Analysis and Statistical Inference

F. Aminravan, F. Bakhtiari-Nejad
Amirkabir University of Technology, Iran

Identification of Noise Emission of a Gear Unit

A. Belšak , J. Prezelj
University of Maribor and University of Ljubljana, Slovenia

17:00 Close of Conference

SESSION 2.1

Keynote Address

WEDNESDAY 12 DECEMBER 2007
14:00 - 15:00

CHAIR

M Moatamedi
University of Salford
UK

Wednesday, 12 December 2007
14:00 – 15:00

Keynote Address

OPTIMAL MULTIPHYSICS AND EXPERIMENTATION IN MICROFLUIDIC CHEMICAL TECHNOLOGY DESIGN

WB Zimmerman and JM Rees
University of Sheffield, Sheffield, UK

Optimization of the dynamics and control of chemical processes holds the promise of improved sustainability for chemical technology by minimising resource wastage. Anecdotally, chemical plant may be substantially over designed, say by 35-50%, due designers taking account of uncertainties by providing greater flexibility. Once the process plant is commissioned, techniques of nonlinear dynamics analysis can be used by process systems engineers to recoup some of this overdesign by optimisation of the plant operation through tighter control (see Bequette, 2003). At the design stage, it should be possible to improve on existing methods by optimally coupling the experimentation with data assimilation into a *multiphysics* model, whilst using such a partially informed, semi-empirical model to predict from parametric sensitivity studies which experiments to run to optimally improve the model. Chen and Asprey (2003) have demonstrated such an approach to optimal experimentation, but limited to a differential algebraic model of the process. Typically, online monitoring has been limited to low dimensional models, e.g. Lane et al. (2003).

Recently Bandulasena et al. (2007abcd) have demonstrated that inverse methods such as data assimilation can be applied to *multiphysics* pde systems with algebraic constraints, a substantially more complicated parameter estimation using finite element multiphysics modelling (Zimmerman, 2006). Craven et al. (2007ab) have shown that parametric sensitivity can be used from such semi-empirical models to predict the optimum placement of sensors to be used to collect data that optimally informs the model for a microfluidic sensor system. This coupled optimum modelling and experiment procedure is analogous to the methodology of Chen and Asprey (2003), but ambitious in the scale of the modelling problem, but also in the scale of the application – a microfluidic device. In general, microfluidic devices are sufficiently easy to fabricate, control, and monitor (see Zimmerman, 2005) that they form an ideal platform for developing high dimensional *multiphysics* spatio-temporal models simultaneously with experimentation.

As chemical microreactors already promise low raw materials wastage through tight control of reagent contacting, improved design techniques should be able to augment optimal control systems to achieve very low raw material wastage. In the presentation, we discuss how the paradigm for optimal modelling and experimentation should be developed and foreshadow the exploitation of this methodology for the development of chemical microreactors and microfluidic sensors for online monitoring of chemical processes. Improvement in both of these areas bodes to improve the sustainability of chemical processes through innovative technology.

References

Bandalusena H C H, W B Zimmerman and J M Rees (Invited Paper, submitted 2007) An inverse methodology for the rheology of a power law non-Newtonian fluid. Submitted to *Mech. Eng. Sci.*

Bandalusena H C H, W B Zimmerman and J M Rees (submitted 2007) Rheometry of non-Newtonian polymer solution by microchannel pressure driven flow. Submitted to *Experiments in Fluids*.

Bandalusena H C H, W B Zimmerman and J M Rees (submitted 2007) Microfluidic rheometry of a polymer solution by micron resolution particle image velocimetry. Part I: Model validation. Submitted to *Langmuir*

Bandalusena H C H, W B Zimmerman and J M Rees (submitted 2007) Microfluidic rheometry of a polymer solution by micron resolution particle image velocimetry. Part II: Inverse methodology. Submitted to *Langmuir*.

Craven T J, J M Rees and W B Zimmerman On pressure sensor positioning in an electrokinetic micro-rheometer device. Part I: Simulation of shear-thinning liquid flows. To be submitted to *Journal of Microelectromechanical Systems.*

Craven T J, J M Rees and W B Zimmerman On pressure sensor positioning in an electrokinetic micro-rheometer device. Part II: Sensitivity, uniqueness and error analysis. To be submitted to *Journal of Microelectromechanical Systems.*

Bequette BW, Process Control: Modeling, Design and Simulation, Prentice Hall (2003).

Chen BH, Asprey SP On the design of optimally informative dynamic experiments for model discrimination in multiresponse nonlinear situations. Ind. Eng. Chem. Res. 42 (7): 1379-1390, 2003.

Lane S.;E. B. Martin;A. J. Morris;P. Gower. Application of exponentially weighted principal component analysis for the monitoring of a polymer film manufacturing process. Transactions of the Institute of Measurement and Control 2003, 25(1), 17-35.

Zimmerman WBJ, Multiphysic Modelling with Finite Element Methods, World Scientific Series on Stability, Vibration and Control of Systems, Vol. 18, Singapore, 2006.

Zimmerman WBJ (ed.), Microfluidics: History,Theory, and Applications, Springer-Verlag-Wien, CISM Lecture Series, No 466, Berlin, 2005.

SESSION 1.4

Flow Modelling and Simulations

WEDNESDAY 12 DECEMBER 2007
15:30 – 17:00

CHAIR

BCE Ewan
University of Sheffield
UK

Numerical Analysis of Fluid mixing in T-Shape Micro Mixer

MUHAMMAD. S. VIRK, ARNE. E. HOLDØ
University of Hertfordshire, UK

Micro fluidic system holds promise for many novel applications within chemistry, biology and medicine. In many of these applications rapid mixing is very important (T. M Squires 2005; J. Green 2007). This has lead to a number of developments of static micro mixers as reviewed by (J. Green 2007). Mixing of fluids at micro scale with in a reasonable time period and length scale is important, but despite a large number of geometries studied, no systematic study on the effect of flow mechanism and geometry has been carried out so far. Aim of this study is to analyze the mixing of two fluids in T-type passive micro mixer under different geometric and flow conditions. Objective of this investigation was to understand the relative effect on mixing by diffusion and dispersion through variation in fluid properties and flow path geometry. For this purpose CFD base numerical calculations are performed using finite element method for laminar flow under steady state conditions. Mixing in T shape static micro mixer is analyzed to study the effect of variation of flow parameters such as inlet velocity, diffusion coefficient and different geometric parameters such as mixing channel width, mixing channel height and angle between inlet channels. Results from 3D numerical simulations show that mixing length scale increases with the increase of inlet velocity and mixing channel width, while there is a considerable decrease in mixing length with the increase of diffusion coefficient and channel aspect ratio. Variation of mixing channel height and angle between inlet channels does not significantly affect the mixing performance.

Numerical Simulation of Boundary Heat Flux during Alloy Solidifcation and its Effect on Natural Convection, Macrosegregation and Microstructure evolution.

K.V. Sreenivas Rao , G. Phankumar, T.S. Prasanna Kumar
Indian Institute of Technology Madras, India

Experiments have been conducted to study the effect of heat flux transients at the metal/mold interface on natural convection, macrosegregation and microstructure evolution of Al-Cu binary alloys. A serial solution of the inverse heat conduction problem (IHCP) is extended for determining multiple heat fluxes at the metal mould interface during casting of Al-Cu alloys in permanent molds. The temperature history of the casting and the mold during solidification is recorded using mineral insulated K- type thermocouples. Computer interfaced data logger (Agilent, model No.34970A) is used for temperature data acquisition. The measured temperatures are used as input to the IHCP algorithm for simulating multiple heat fluxes at the boundary. The obtained heat fluxes are used as boundary conditions for the casting simulation. The effect of heat flux transients on thermosolutal convection and macrosegregation is studied. It is observed that the measured heat flux is a function of alloy composition, thermo physical properties of the mold material and the mould coating. The effect of heat diffusivity of the mold on interfacial heat flux and cooling rate is analyzed. The cooling rate decreases with increase in interfacial thermal resistance at the metal/mold interface and affects macrosegregation and microstructure formation. Macrosegregation increases with the decrease in cooling rate and increase in the overall composition of the alloy.

Keywords: IHCP, Boundary flux, Thermosolutal convection

STUDY OF AIR-FLOW AND HEAT TRANSFER ON A LARGE-SCALE ROLLER-KILN VIA COMPUTATIONAL SIMULATION AND EXPERIMENTAL PROTOTYPE

A.Chai, A.Vakhguelt
Swinburne University of Technology, Malaysia

In this paper, the comparison of both computational simulation and experimental results of a large-scale roller-kiln is presented. The roller-kiln is a high-temperature furnace utilized in the ceramic-tiles manufacturing sector of the Malaysian industry. The roller-kiln has 20 side-installed liquid petroleum gas (LPG) burners, capable of producing heated air-gas at a temperature range of 650°C to 1200°C for the baking process of the ceramic-tiles. A commercially available computational fluid-dynamics (CFD) solver, Star-CD, was employed for the simulation of the air-flow circulation and heat-transfer within the main baking chamber of the roller-kiln. The modeling and cell-meshing were also performed using Star-CD, where boundary conditions and thermophysical properties were established according to the actual roller-kiln and LPG burners operating conditions and parameters. CFD results for both air-flow circulation and heat-transfer are shown in periodic contour-plots and particle-tracking streamlines. Iso-surf plots were also generated to illustrate the transfer of heat within the baking chamber, with the pathway originating from the burners. The CFD results also provided a better observation to the heat-transfer phenomenon and areas where further improvement can be performed on the roller-kiln. A scaled-down experimental prototype was later constructed to replicate the roller-kiln. Boundary conditions and thermophysical properties were also set-up according to the actual conditions of the roller-kiln. Results were obtained for the circulation of air-flow and heat via data-acquisition using vane air-flow sensors, thermal sensors and thermocouples. The results for both the CFD simulation and experiment were then compared and studied. Both set of results showed that heat-transfer within the roller-kiln was homogeneous. Air-flow was found to be consistent at similar locations within both CFD model and experimental prototype. Both set of results showed concurrence. This implied that the boundary-conditions and parameters were established appropriately and that the results were presented with a good level of accuracy.

Keywords: computational simulation, ceramic-tiles, modeling, cell-meshing, prototype

Understanding in Multiphysics of the Unsaturated Flow in Porous Media

Y. Wang, M. Moatamedi
University of Salford, UK

Fluid flow in unsaturated porous media has a great interest in a wide range of science and engineering subjects, such as environmental, material science/engineering; hydrology; soil physics; soil mechanics and oil/gas engineering. Due to the nature and complexity of porous media, unsaturated flow is an inherently complex science that requires a multidisciplinary knowledge including physics, chemistry, mathematics, computer science and fluid dynamics to ascertain answers to contemporary relative issues. Because the voids in unsaturated porous media are generally filled with both liquid and gas, the flow parameters depend on transient phase states. For example, unsaturated hydraulic properties vary with the saturation of soils. The saturation refers to the ratio of the total voids in soils filled with water.
Water retention characteristic is an important constitutive feature of porous media. Meanwhile it is a basic requirement for unsaturated flow modelling. It defines the relation between matric potential or capillary pressure and water saturation. It is usually expressed using water retention curves (WRCs). The mechanism of water retention characteristic in unsaturated porous media is still not well understood. As a result, most of the existing models are not satisfactory in performance. In this work, we try to investigate this topic from a point of multiphysical view. We'll analyse the individual physical and chemical behaviour of the coexisting two fluid phases and the interfacial interactions between the three phases involved. The analyse derives out a function which represents the water retention characteristic. This function demonstrates the important effect of the specific surface area of the porous media on their WRCs. Its mathematical analysis is found to be in agreement with experimental observations. Finally, a model directly based on this function has been proven to be more accurate and stable than most of the existing main models when used to fit the WRCs of porous media.

SESSION 2.1

IMPACT DYNAMICS

THURSDAY 13 DECEMBER 2007
09:00 - 10:30

CHAIR

S Chynoweth
Shell Global Solutions
UK

Influence of Projectile-Barrell Interactions on Launch Dynamics

M.M. Chen
Army Research Laboratory, USA

A novel approach is proposed to assess the effects of gun barrel centerline variations on projectile launch dynamics. Four fundamental shape functions were created to model the deformation of gun barrel geometry. The levels of the shape values were determined in accordance to reasonable manufacturing tolerance. With the technique of design of experiments, a full factorial design was adopted to simulate an array of flexed gun tubes. The simulations of the design set constituted a good representation of experimental gun barrel centerline profile.
In-bore dynamic analysis of a projectile system subject to the simulated barrel shapes was performed. The projectile responses at the muzzle including transitional and rotational velocities, which are significant to fly-out stability and accuracy, were obtained. To evaluate the performance, certain aerodynamics characteristics, such as the first maximum yaw angle and aerodynamic jump, were calculated. In addition, analysis of variance method was employed to identify the sources of variability and to determine the significance of each barrel shape function. Response statistics were also derived to assess the influence of projectile-barrel interactions. The results should serve as a guideline to predict launch conditions in response to normal barrel centerline deviations.

Key words: Projectile Launch Dynamics, Barrel Centerline Variation, Design of Experiments, Analysis of variance

Pressure Vessel made by Free Forming using Underwater Explosion

H. Iyama, H. Maehra, Y. Hidaka, S. Itoh, Yatsushiro
National College of Technology and Kumamoto University, Japan

The underwater shock wave is generated by underwater explosion of explosive. We produced the pressure vessel using this shock pressure. This technique is called explosive forming. But, we can control the underwater shock wave and get the appointed shape of metal by some times explosive forming. In this paper, we present the experimental results and some numerical simulation results.

Keywords: Pressure vessel, Explosive forming, Underwater explosion

Deformation of Egg-Box Panel subjected to Compressive Load

S. Nowpada, E. C. Chirwa, P. Myler
The University of Bolton, UK

Aluminium Egg-Box panels have been subjected to Quasi-static Compressive Load. The resulting Load-Deflection and Stress-Strain curves were used to assess the energy-absorbed, pattern and mechanism of collapse. Theoretical analysis of the structure's deformation was carried out and ANSYS, LS DYNA were used for Numerical Analysis. Thus the assessment of crashworthiness of the panel will be discussed.

In the fifteen EU member states there are approximately 42,000 reported deaths and 1.5 million people injured every year as a result of road accidents. To alleviate this problem countermeasures based on casualty reduction potential are being undertaken in pursuit of structural crashworthiness.

The manner in which the impact load is transmitted through the vehicle system in an accident directly affects the magnitude of damage to the occupants. A crashworthy system is essential to increase the probability of survival of the passengers by minimizing the forces and accelerations that are likely to be experienced by them. Material technology is playing a key role in making automobile vehicles safer for both pedestrians and passengers, by acting as energy absorbers to cushion the impact in an accident. Research studies reveal that materials with low density or light mass are proving to be very good alternatives. A new class of such energy absorbing material is the "Egg-box" Aluminium panel. This is a 'sacrificial' energy absorber which can dissipate kinetic energy in a controlled manner whilst undergoing plastic deformation.

The quasi-static load-deflection curves obtained by testing the panels under compressive load give an estimate of the transmitted force levels and their energy absorbing capacity. The pattern of deformation will be studied and numerically analysed using ANSYS and LSDYNA. The study will thus be validated and discussed.

Keywords: Quasi-static Compressive Loading, Structural Crashworthiness, Plastic Deformation.

Buckling Failure of Thin Walled Cylindrical Shells Under Axial Compression

H. Ullah
NESCOM, Pakistan

Light weight thin walled cylindrical shells subjected to external loads are prone to buckling rather than strength failure. In this paper, buckling investigation of thin walled cylindrical shells under axial compression is presented. Buckling failure is studied using analytical, numerical and semi empirical models. Analytical model is developed using Classical Shell small deflection theory. A Semi empirical model is obtained by employing experimental correction factors based on the available test data to the theoretical model. A finite elements model is built using ANSYS FEA Code for the same shell.
Finally, the different results obtained using the three analysis methods are compared. The comparison reveals that analytical and numerical linear model results match closely with each other but are higher than the empirical values. To investigate this discrepancy, non linear buckling analysis with large deflection effect, is carried out. The effect of geometric imperfection is also studied through a nonlinear model. These nonlinear analyses show that the effects of nonlinearity and geometric imperfections are responsible for the difference between theoretical and experimental results. The effect of shell thickness, radius and length variation on buckling load and buckling mode has been also studied.

Keywords: Buckling, Semi empirical, ANSYS

SESSION 2.2

SHOCK AND EXPLOSIVES

THURSDAY 13 DECEMBER 2007
11:00 - 12:30

CHAIR

Z Ren
University of Maribor
Slovenia

The Numerical Analysis of Food Processing using Underwater Shock Wave

N. Okamoto (1), A. Oda (1), M. Moatamedi (2), S. Itoh
(1) *Kumamoto University, Japan,* (2) *University of Salford, UK*

In recent year, various processes and techniques using high energy pulsed power of short duration are in practical use. Here, we focused non-thermal processing of food using underwater shock wave. This process has various merits such as a short processing time and no nutrients reduction due to no thermal processing.
When we try to put this food processing into a practical use in the future, a suitable food processing container have to be developed. In order to shorten the time for the development of the suitable food processing container, it will be very important the investigation of container by not only the experiment but also the numerical analysis. However, the shock parameters for various foods are still not be unknown, so that an useful numerical analysis has not been done until now. In this study, first, we estimated the Us-Up (shock wave and flow velocity) relation to make the equation of state analyze of apple. Then we will show some numerical results for shock loading on apple by LS-DYNA-3D
The experiment to determine the relation Us-up for apple, were carried out using the high-speed image converter camera and high speed video and the explosive experimental facilities developed in our laboratory.

On the Destruction of the Cell Wall of Plants and its Mechanism by the Shock Wave

Takemoto, K. Kuroda, H. Iyama, S. Itoh
Yatsushiro National College of Technology and Kumamoto University, Japan

The shock wave transmitted at the speed that exceeds speed of sound generates the expansion wave by the density difference. The expansion wave causes negative pressure on the density difference side, and causes the spalling destruction. This phenomenon is caused also on the surface of the cell wall of the plant. The cell of the plant has the cell wall to maintain the shape. This is a big difference between the animal cell and the plant cell. That is, the plant has all big density differences in the cell structure. Therefore, the spalling destruction on the surface of the cell wall can spread to the entire cell tissue of the plant by the shock wave. It is possible to apply it to the extraction of the cytoplasm etc. by the cell wall destruction by using this phenomenon. The mechanism to which walls of the plant cells are destroyed by the shock wave is described in the experiment and the comparison with the numerical simulation in this research.

Keywords: Shock Wave, Plant cell, Extraction, Numerical Analysis

Basic Study on Promotion of Thawing Frozen Soil by Shock Loading

T. Watanabe (1), H. Maehara (1), S. Itoh (2)
(1) National Fisheries University, Japan, (2) *Kumamoto University, Japan*

The aim of study is to confirm a new technique that can crush the frozen soil and/or ice block using underwater shock wave generated by the underwater explosion of explosive. This technique can lead to the earlier sowing, which can have the larger harvest because the duration of sunshine increases. Especially, in Hokkaido prefecture, Japan, if the sowing is carried out in April, we can expect to have 150% of harvest in the ordinary season. This technique is effective against the cold regions for example, Korea, China, Mongolia, Russia, Norway, and Sweden, etc. At first, we carried out experiments using a detonating fuse and ice block. The process of ice breaking was observed by means of a high-speed camera. In order to check about that influence we tried to give an actual frozen soil a shock wave. We carried out numerical simulation in order to explain the influence of surface water on the frozen soil layer in the case of promoting frozen soil thawing by the shock loading. This result corresponded to a result of an experiment qualitatively.

Keywords: Frozen soil, Shock loading, Thawing

Numerical Analysis of Detonation Phenomena using PBX High Explosives

K. Kuroda (1), M. Otsuka (1), E. Hida (1) M. Moatamedi (2), S. Itoh (1)
(1) Asahi Kasei Chemicals Co., Japan, (2) *University of Salford, UK*

Detonation products of explosives are described by various types of equations of state. Jones-Wilkins-Lee (JWL) equation of state is widely used because of its simplicity. We obtained previously unknown JWL parameters of PBX high explosives through the method of characteristics applied to the cylindrical configurations. To verify the JWL parameters, high speed video observation experiments were performed. Numerical analysis using LS-DYNA3D provided good agreement with results from the optical observation experiments.

SESSION 2.3

MICROMECHANICS

THURSDAY 13 DECEMBER 2007
13:30 – 15:00

CHAIR

S Itoh
Kumamoto University
Japan

Exact Code Scaling

M. Antal, Makai
BME Institute of Nuclear Techniques, Hungary

Nuclear analysis codes are often validated against measurements carried out in a mock-up. Then, the measured values are transformed from the mock-up to the real geometry. That procedure is called code scaling. We address the questions: are there mock-up geometries allowing for an exact code scaling? Are there geometries, such that the solution of a boundary value problem in one can be transplanted into the other exactly? The answer is positive.
Consider the solution of a boundary value problem in a discretized volume *V*, which is constructed from copies of an elementary unit, called tile *t*. A tile is a finite domain of the space bordered by a finite number of straight lines. Copies of *t* are obtained by reflections through its sides. We describe *V* by algebraic methods: a group G_V and a graph $_V$ are assigned to *V*. G_V is a finitely presented group; its generators are associated with a permutation representation of the edges. If one can find two discretized volumes with identical group G_V, there may exist a simple transplantation rule. Using computation methods, elaborated in the last years, one can find an algorithm to find equispectral discretized volumes. The Sunada theorem guarantees the existence of equispectral discretized volumes. Algebraic description of *V* is useful in studying the solution. When *V* consists of *N* copies of *t*, the solutions in the copies are collected into a vector $\mathbf{\Phi}(x) = \left(\Phi_1(x),\ldots,\Phi_N(x)\right)$, and that vector is expressed in terms of the solutions along the internal edges $\vec{f}(x) = \left(f_1(x),\ldots f_K(x)\right)$ as

$$\mathbf{\Phi}(x) = \mathbf{Q}\mathbf{v}(x)\,,$$

where

$$\mathrm{v}_i(x) = \sum_{k=1}^{K} \int_{e_k} S_{ki}(x,x')\, f_k(x')dx'\,.$$

Here function $v_i(x)$ is a function over copy *i* of tile *t*, S_{ki} is the Green's function of copy *i* of tile *t*, and $S_{ki}\neq 0$ only when internal edge *k* is the boundary of copy *i* of tile *t*. Matrix **Q** depends on the structure of *V* and is obtained from the algebraic description.
The presented method separates the effect of the geometry (in **Q**) from the equation to be solved ($v_i(x)$) and opens new ways to find equispectral volumes by numerical techniques. In the presentation, methods and numerical examples will be given to transplant measurements.

The Over-Barrier Resonant states and Multi-Channel Scattering by a Quantum Well

A.F.Polupanov, V.I.Galiev, A.N. Kruglov
Institute of Radio-Engineering & Electronics of the Russian Academy of Sciences, Russia

Using the ‘recurrent sequences‘ procedure [1-4] we demonstrate, that one or several resonant hole states, related to the absolute reflection at hole scattering by a quantum well, exist in the energy range where only the heavy hole may propagate over barriers in a quantum-well structure. Hole states are described by the Luttinger Hamiltonian. The qualitative behaviour of the over-barrier scattering and resonant states is the same at variation of the quantum well potential shape and depth, however lifetimes of resonant states depend drastically on all parameters.

1. A.F. Polupanov, Fiz. Techn. Poluprovodn. 1985, 19, 1647.
2 V.I. Galiev, A.F. Polupanov, I.E. Shparlinski, J. Comp. Appl. Math., 1992, 39, 151 .
3. V.I.Galiev, A.F. Polupanov, J. Phys. A .: Math., Gen. 1999, 32, 547.
4. V.I. Galiev, A.N. Kruglov, A.F. Polupanov, et al. Semiconductors, 2002, 36, 546.

Keywords: recurrences, scattering, quantum wells

Study on Low Velocity Detonation Phenomena in Nitromethane

A. Osada, H. Hamashima, Y. Kato, S. Itoh
Kumamoto University and AIST and Nippon Koki Co Ltd, Japan

In detonation of an explosive, there are two forms, high velocity detonation (HVD) and low velocity detonation (LVD). In a liquid explosive, the highest pressure of LVD changes with explosive and conditions, is a few GPa and has destructive power equivalent to HVD. It is important also about security to get to know the actual condition of LVD. Moreover, it is important that the performance of explosives is grasped completely to control HVD and LVD and deflagration and to demonstrate an expected function.
Nitromethane (density 1131∼1137kg/m3) is typical liquid explosive, and since it is homogeneous and isotropic, compared with a heterogeneous solid explosive, the character is simple. Although it is the energy substance which has the explosion power exceeding TNT since it is comparatively insensible, it is used as an organic composition material in an industrial field, only as an explosive. Since the ignition point is comparatively high with 44 ,it is used as a solvent of a paint.
So, in this research, in order to grasp HVD and the LVD characteristic of NM of being suitable for observing air bubbles and a shock wave because of its clearness, we observe optically using a high speed camera. The mechanism of cavitation bubble generating by the precedence shock wave which is the feature of a LVD phenomenon is solved based on the data.
The result of research, generation of the cavitation which is an important factor for LVD generating was showed by optical observation. The action of a precedence shock wave and detonation wave was also observed, and each velocity was calculated with the streak photographs and the framing photographs. This result showed that the distance which detonation wave maintains became short as Gap length became long, incidence pressure became low.

Fission Models and Odd Even Effect in Charge in Low-Energy Fission

G. Medkour Ishak Boushaki, M. Asghar, N. Boucheneb, P. Geltenbort, G. Barreau, M. Allab
Institute of Physics, Algeria

The problem that continues to confront the low energy nuclear fission is to understand the nature of dynamics of this process which shows up when the fissioning nucleus moves irreversibly, in a multidimensional deformation space, from the last barrier towards the scission point. To approach this dynamics, we study the variation of the odd–even effect in charge P as function of the fission fragment kinetic energy in the case of 235U(nth, f). We also consider the divers types of available fission data on the thermal-neutron induced fission of nuclei ranging from 229Th to 249Cf. Qualitative comparison is made of the experiment data with the predictions of the different models of nuclear fission.
This analysis indicates that the band-of-fission model of Nörenberg is compatible with the existing experimental evidence and the saddle-to-scission fission dynamics seems to be mostly collective in nature. The quasi-particle excitations, hence pair breaking, seems to be produced during a fast neck–rupture at scission.

Keywords: fission, odd even effect in charge, fission dynamics

SESSION 2.4

Electrophysics

THURSDAY 13 DECEMBER 2007
15:30 – 17:00

CHAIR

AE Holdo
Coventry University
UK

Multiphysics Simulation of a Micromirror Device

S. Kini, K. Shah, M. Megahed
ESI Group, Germany

With the advancement in simulation technology and computational capability, Multiphysics simulations are becoming increasingly affordable and thus more and more popular during the design process. The advantage of such simulations over traditional uncoupled calculations, such as CFD-only or Stress analysis-only, Electric-only, etc., is the fact that effects of various physical phenomena and their interaction can be incorporated into one single simulation.
One such multi-physics problem, occurring in the Projection TV industry, is the electric actuation of a micromirror device. Here a micromirror is made to move about a hinge using an electric bias voltage so that it reflects light in a desired fashion. The movement (bending, torsion, etc.) of the mirror assembly leads to stresses within the part that need to be kept in check. The fact that these micromirrors operate in air, requires that the fluid flow also needs to be accounted for in order to determine the damping influence on the mirror motion. This makes it necessary to use fully coupled multi-physics simulations in order to capture the full complexity of this device.
The primary objective of this paper is to describe the multi-physics simulations conducted on a generic micro mirror. The simulations account for the electrostatic actuation, the stresses within the device components and compute the fluid damping effects on the system. This work also simulates contact between the mirror and the landing posts placed to prevent shorting with the bias electrodes. The mirror motion as well as the deformation of different components is accounted for using grid deformation algorithms.
The secondary objective relates to the fact that, in reality, there are a number of parameters involved in the design of such devices and typically a significant number of simulation runs will be required to tweak and perfect all the parameters. Towards this end this paper will present how python scripting can be used within the simulation framework to efficiently run a series of cases with parameter variations thus reducing man-time and turnaround time and allowing this simulation process to fit into the design schedule.

Keywords: Micro-mirror, Electrostatics, Fluid Flow, Stress Analysis, Fluid-Structure Interaction (FSI), Fluid Damping, Multiphysics, Contact Analysis, Scripting, Parametric Analysis

In-Orbit Attitude Actuation Using Solar Panels

R. Varatharajoo, T.M. San
University Putra, Malaysia

The photovoltaic solar power generation method is employed in spacecraft. In small satellites, the solar cells are body-mounted, and therefore, the power generation capability is governed by the orbital motion. On the other hand, most high-end space missions have quite a significant power requirement. In this regards, the use of solar array drive mechanism becomes desirable in order to constantly track the sun. The solar panels are constantly rotated using DC motors. As a consequent, the internal attitude disturbance torque is simultaneously generated, which has to be rejected in order to maintain the spacecraft attitude accuracy. In this paper, a specific technique is developed to wield the internal disturbance torque for attitude control tasks instead. Hence, this work will be the maiden work towards integrating the attitude control and the solar tracking tasks, forming a combined attitude and solar tracking system. The feasibility of this concept for spacecraft is proven and eventually the combined concept is validated. A technical proof is presented corresponding to the end-to-end system demonstration. The investigation starts with the determination of the solar tracking constraints. Then, the mathematical models describing the attitude and solar tracking are established, and eventually the onboard architecture is implemented. The numerical treatments using Matlab™ were performed to evaluate the developed onboard architecture. The simulation results are discussed especially from the attitude standpoint. The integrated system complies very well with the reference mission requirements. Hence, employing such an integrated system onboard spacecraft would benefit the mission itself, e.g., performance enhancements, mass and volume savings, etc. Finally, this end-to-end system demonstration indicates that the integrated attitude and solar tracking concept is judiciously a feasible option and it can be easily implemented in spacecraft

Keywords: Spacecraft Attitude Tracking, Spacecraft Power Tracking, Combined Systems

Circulation of Slag and Molten Metal in a Submerged Arc Furnace

I. Mc Dougall, J.-H. Grobler
CSIR, South Africa

A finite element model was generated of a 48MVA AC submerged arc furnace. The model included the electrodes, raw material, slag and molten metal. The electric current distribution and resulting magnetic fields were calculated in the slag and molten metal of the furnace using ANSYS/Multiphysics. A CFD model of the furnace was generated, which included both the slag and molten metal. The magnetic flux density obtained from the finite element analysis was used as input to a CFD model of the furnace, which included both the slag and molten metal. The circulation pattern in both the slag and molten metal as a result of the magnetic field and the electric current was calculated using FLUENT. The circulation was found to be relatively insignificant, as compared to thermally-induced circulation.

Keywords: submerged-arc furnace, circulation, MHD, electric current dictribution

Gamma Radiation Effect on Enhancement VMOSFET

S. A. Hayat, M.I. Sabir, R.Raza
COMSATS Institute of Information Technology, Pakistan

MOSFET being the surface device is sensitive to radiation. The effect of γ-radiation on turn on voltage in case of P-channel enhancement vertical MOSFET has been measured. The devices showed a shift in measured turn on voltage and Transconductance. The effects may be due to radiation induced charge build up in oxide and interfacial regions.

Keywords: MOSFET, turn-on voltage, Conductance, γ-radiation, Transconductance (gm)

SESSION 3.1

SOFTWARE DEVELOPMENTS

FRIDAY 14 DECEMBER 2007
09:00 – 10:30

CHAIR

T Morris
NAFEMS
UK

NAFEMS: Engineering Analysis and Simulation

T. Morris
NAFEMS, UK

Engineers rely on computer modelling and simulation methods and tools as vital components of the product development process. As these methods develop at an ever-increasing pace, the need for an **independent, international authority** on the use of this technology has never been more apparent. NAFEMS is the only worldwide independent association dedicated to this technology.

Companies from numerous industries and every part of the globe have invested heavily in engineering technologies such as FEA and CFD.

But -

How do they ensure they get the best return from their investment?
How do they develop and enhance their capabilities?
How do they know they are using the technology in the most effective way?
NAFEMS is uniquely placed to help answer these questions.

NAFEMS is a vendor neutral, not-for-profit membership association of more than **800 companies** from all over the world. Members range from major corporations such as Boeing through mid-sized organizations such as JCB, to small-scale engineering consultants.

Multiphysics Modelling and the COMSOL Methodology

K. Howard
Comsol, UK

Over the past thirty years modelling methods, such as finite elements, have been adopted by the engineering and scientific community as the accepted way of simulating complex physical behaviour. Indeed it would be impossible to develop many of today's products in acceptable timescales without recourse to 'software prototypes'.
However, most proprietary modelling solutions have focused on 'single physics'- such as electromagnetics, structural mechanics and fluid flow. These schemes can be adapted to cater for some aspects of Multiphysics modelling, for example by adding heat transfer to fluid flow, but in doing so they carry inherent limitations. And yet many phenomena that need to be modelled are truly Multiphysics in nature.
COMSOL has built, from its very beginnings, a technology that makes it possible to solve for any, arbitrary set of non-linear equations that can be coupled in a single solution. Moreover the technology allows users free access to all the equations making it easy for users to adapt the solution to any equations they care to define. In this way COMSOL Multiphysics provides an easy to use and robust environment in which to build true Multiphysics simulations.
This presentation will provide a background to the underlying architecture necessary for Multiphysics modelling and how this has been implemented in COMSOL Multiphysics.

ANSYS Multiphysics - An Integrated and Unified Virtual Prototyping Tool Kit for Industry and Research

B. Miller
Wilde FEA Ltd, UK

As companies strive to produce innovative and higher quality products, the simulation of individual physics is no longer adequate for the design of many products. Multiphysics simulation allows engineers and designers to evaluate their designs operating under real-world conditions. The ANSYS Multiphysics solution allows engineers and designers to simulate the interaction between structural mechanics, heat transfer, fluid flow, acoustics and electromagnetics. This technology is provided within a CAD-integrated, unified simulation environment, minimising the effort required to mesh and pre-process the models, together with efficient results postprocessing.
Two solution techniques for solving coupled-physics problems have been developed for the ANSYS Multiphysics product - directly coupled- field elements and an in-house multi-field solver - depending on the application. The directly coupled-field elements provide both matrix and load vector coupling for solving problems such as Joule heating and electrostatic actuation. Alternatively, the multi-field solver approach provides robust implicit sequential coupling for solving problems such as fluid-structure interaction and induction heating. Together, the two solution techniques have enabled engineers and researchers to solve an extremely broad range of industry applications. In particular, complex devices such as thermoelectric coolers, accelerometers, inkjet printers, integrated circuits, solenoids and electric motors are commonly developed using this technology.
Based near Manchester, Wilde FEA Ltd are an ANSYS channel partner providing FEA and CFD software, training and consulting. This presentation will provide an overview of the ANSYS Multiphysics solution techniques and range of capabilities, including a snapshot of typical simulations performed by users today.

PHYSICA: PHYSICA Approach to Challenging Multiphysics Problems

L. Marks
Physica, UK

At the heart of most manufacturing processes lie a range of physical and chemical phenomena that involve a number of complex interactions. Fluid-structure interaction, aero-acoustics, magneto-hydrodynamics with heat transfer and solidification/melting, and thermally driven material defects are but a few of the most commonly found examples in manufacturing process industries.
PHYSICA is a new generation of simulation software that is specifically designed to tackle phenomena associated with manufacturing processes. The following list contains examples of industrial applications that have been modelled using PHYSICA:
Casting and solidification processes
Electronic manufacturing processes
Granular Flow
Welding
Forming processes (e.g. extrusion)
Primary metals processing (e.g. direct smelting, heap leaching)
KEY FEATURES OF PHYSICA
State-of-the-art simulation technology
Powered by multi-physics algorithms
Tackles diverse phenomena under a single unified environment
Efficiently handles three dimensional complex geometries
Parallel implementation available for faster solutions

SESSION 3.2

HEAT TRANSFER AND THERMODYNAMICS

FRIDAY 14 DECEMBER 2007
11:00 – 12:30

CHAIR

A Tehrani
Serco Assurance
UK

Virtual Cooking

C. Catalogne, J-Y. Noel, L. Padovan, G. Elger
Electrolux Major Appliances Europe, Italy

Computer Aided Engineering presents a huge potential to analyse the performance of a system and provide a significant help for the development engineer to understand the functioning of its product and to run “what if” scenarios. In dependence of the simulation targets, functioning of single components, subsystems or a complete product can be analyzed.
We developed a CFD simulation model to simulate air flow, temperature distribution and heat transfer in household ovens to predict cooking performance, i.e. food browning. Our virtual oven includes the three main heat transfer channels, i.e. radiation, convection and conduction. The convection fan is modelled using its real geometry as rotating mesh. Care was taken to model the important geometric features like the fan cover and the heating elements appropriately but unnecessary geometric details were simplified to keep the mesh feasible. Different load settings, i.e. different numbers of trays, the brick or pastry stripes are implemented into the model.
The heating power input of the heating element is controlled by a virtual temperature sensor as in the real product. With this temperature control we are able to simulate the heat up phase and also the energy consumption based on the test procedures of the CENELEC, European Standard EN 50304 for energy consumption.
The model was validated using flow visualization like helium filled soap bubbles and flow velocity measurement techniques like LDA. For temperature validation the temperature was measured at various positions in the oven, i.e. on the trays and the load.
For beneficial use of the virtual oven within product development, it was important to develop an efficient modelling strategy, i.e. a workflow was established starting from CAD, via mesh creation, physical boundary condition setting, solving and post-processing. The time for model generation from initial CAD data to post-processed results was decreased from 3 month to 1 week for a typical oven geometry (cavity, heating element, fan cover, load).
The paper focuses on the description of the simulation methodology and the validation of the CFD model. The application and its beneficial use will be demonstrated by examples.

Numerical Prediction of Indoor Temperature Stratification

K. Chow, A. E. Holdø
University of Hertfordshire, UK

In many buildings, displacement ventilation is used so that contaminants can be separated from occupied space [0,0]. A fundamental characteristic of displacement ventilation is a layer of stratified fluid that separates the occupied zone from the contaminated zone, marked by a density gradient usually due to temperature differences in the two zones. Because buoyancy is a dominant effect in these situations, natural rather than forced ventilation is commonly used.
This paper presents a series of simulations studying the prediction of steady-state stratification in a room ventilation case. Following on from work performed by Iial-Awad [0], numerical simulations coupled with a 2-equation turbulence model were employed to predict the steady-state time-averaged temperature distribution and stratification layer height in a room subject to two momentum sources at different temperatures. The room has been simulated with different levels of idealism to examine the effect boundary condition assumptions have on the predictive accuracy of the simulation compared to the work of Iial-Awad [0].
It was found that the assumption of adiabacity of the room caused the predicted vertical temperature profile to be highly idealistic, with the flow stratified into two discrete layers across a sharp interface. The addition of heat loss boundary conditions and thermal modelling causes the predicted temperature profile to more closely match that produced in the original experimental work.

Keywords: RANS, ventilation, stratification

1. Linden, P. F., (1999), “The Fluid Mechanics of Natural Ventilation”, Annual Review Fluid Mechanics, **31**, 201-238
2. Holdford, J. M., Hunt, G. R., (2003), ”Fundamental atrium design for natural ventilation”, Building and Environment, **38**, 409-426
3. Iial-Awad, A. S., (2006), “Stratified Flow in the Built Environment”, Ph.D Thesis, University of Hertfordshire

A Numerical Investigation and Fluid-Dynamical and Thermo-Chemical Operative Conditions Concurring in Metal Deposition on Post-Combustion Burners

C. Giuliano, C. Salvatore, C. Luigi, P. Giuseppe
University of Catania, Italy

One of the most recent typology of power plants is based on the combined cycle technology: a turbogas provides exhaust high-temperature gas (Turbogas Exhaust Gas, TEG) to a Heat-Recovery Steam Generator (HRSG) of a steam plant. In most cases the HRSG are also equipped by after-burner sections in order to increase the power output of the steam cycle. In many applications, the after-burner system is formed by an array of combustors, which are mounted on fuel pipes and uniformly distributed in a section of the duct (duct-burners) connecting the gas turbine with the HRSG. From an applicative point of view, these components are often subject to significant thermal stresses and even cracks or permanent deformations due to very high temperature values close to their surfaces. Otherwise, when used fuels contains metals, deposition of metallic composts could be detected on the after-burner surfaces and on the supply manifold, which causes a long-term destruction of the pipe due to the local concentration of thermal stresses. The deposition of metals in the injection nozzles also causes their occlusion, which requires unacceptable stop frequency for plant maintenance. In this study a multi-physical FEM based software is exploited in order to simulate reacting flows for the above discussed technological systems. Fluid-dynamical analysis is performed using a k-e turbulence model, while six transport-diffusion equations are solved in order to simulate the reacting flow of CO, H2, CH4, O2, CO2 and H2O, respectively. Reaction enthalpy is then considered in the energy balance in order to solve thermal field of the considered computational domains. The results are obtained for several chemical compositions of the supplied fuel and for several values of the fuel mass flow rate incoming inside the control volume. Three-dimensional simulations principally show as an accurate design of the combustors array is strictly needed in order to avoid incoming TEG to flow close to the injection nozzle of fuel. Otherwise, off-design operative conditions concerning mass flow rate of fuel also determinate regrettable high temperature values on the after-burners manifold.

Modeling the Thermal Behavior of Fluid Flow inside Channels Using an Artificial Locally Linear Neuro-Fuzzy Approach

A. Hashemian, T. Babaie, C. Lucas
Sharif University of Technology, Iran

In classic fluid dynamics, using the three main equations of mass, momentum and energy conservation, one might determine the velocity and temperature field via numerical methods. Numerical simulations and algorithms have shown good performance in most cases but they need powerful processors and large memories while the solution procedure might be very complicated and time-consuming one, if not impossible.
Artificial learning methodologies, e.g. biologically motivated learning algorithms, seem to be adequately powerful to be used in proposed problem. Various numerical techniques have already been demonstrated for modeling of nonlinearity. In recent years, the most successful approach used for modeling and prediction of nonlinear systems has been neural networks, Neuro-fuzzy models and biologically motivated learning algorithms, e.g. genetic programming, evolutionary algorithms and reinforcement learning. Among these, neural network models have yielded better accuracies due to their nonlinear mapping capabilities.
In the present research, we study the quasi periodic and unpredictable behavior of fluid flow in rectangular and corrugated channels, considering the two dimensional Navier-Stokes and energy equations. Firstly, we obtain the velocity and temperature fields of a specified rectangular channel via a finite volume CFD method. The numerical data would be arranged as a set of training data for the neural network. Then we use a Locally Linear Neuro-Fuzzy method based on radial basis function (RBFs) to extract the simple and accurate dynamic model from the achieved numerical fields. The result is then applied into a corrugated channel with similar boundary conditions. In this way, we utilize an incremental learning algorithm named model tree algorithm to learn our proposed Neuro-Fuzzy model. We show that by extracting the nonlinear dynamics of the equation using locally linear approach in neural network with fuzzy interface, a smart neural model is obtained. Finally, the model helps us to determine the temperature field in the corrugated channel.

Keywords: channel flow, neural network, temperature field

SESSION 3.3

FLUID STRUCTURE INTERACTION

FRIDAY 14 DECEMBER 2007
13:30 – 15:00

CHAIR

M Souli
University of Lille
France

Computational Simulations of a Spacecraft Water Landing and the Water Impact of a Sphere

M. Vesenjak (1), Z. Ren (1), M. Moatamedi (2)
(1) University of Maribor, Slovenia, (2) University of Salford, UK

The paper presents computational modelling of water impact. A spacecraft water landing and sphere water impact are considered as examples, which have a strong relevance in aircraft water impact crashworthiness and have been recently identified as an area requiring further research. Available numerical techniques in the framework of the finite element method have been studied for the analysis of the underlying transient dynamic, fluid-structure interaction problem. In scope of this research the Arbitrary Lagrange-Eulerian and the Smoothed Particle Hydrodynamics methods have been used to simulate the behaviour of the fluid (water) under impact conditions. The evaluated accelerations and velocities of the impacting objects were validated with experimental measurements and compared with results of available analytical methods. The computational results show strong potential for use of developed computational models for analyses of water impact problems. In both of the problems addressed, it is shown that the development, evaluation and validation of numerical modelling techniques accounting for the fluid-structure interaction can benefit the future design, certification and accident investigation of water impacting objects.

Keywords: Water impact, Computational simulations, Fluid-structure interaction, ALE, SPH

On the Coupled Dynamics of Wave-Seabed Interaction

S. Williams
The University of Sydney, Australia

Fluid mud layers develop close to the seabed either by fast deposition of suspended particulate matter or by the fluidisation of recently deposited sediment layers. Some common marine pollutants such as trace metals and pesticides are known to settle and accumulate in such muds. The mud layer may fluidise due to wave action and this process is known to create layers of mud with properties which are dependent on the magnitude of the induced oscillatory strain.

The objective of this study is to investigate the propagation of a water wave above a stratified seabed, where the bed is considered to have layers with different, yet constant, density and viscosity. The upper layer of the bed is considered as a dense and viscous fluid mud, which is underlain by a porous domain.

The height of the surface wave is specified and the initial characteristics of the forced interfacial wave and the depth of the fluid mud layers are determined as a function of the imposed oscillatory wave characteristics. The viscous attenuation of the wave due to the combined effects of the wave interaction with the porous seabed, damping within the fluid layers, and damping at the interfaces is considered. An Arbitrary Lagrangian-Eulerian finite element method is used along with the incompressible Navier-Stokes equations in two-dimensions to describe the fluid motion within the water column and the more viscous homogeneous fluid mud.

The key focus of this work is on coupling the dynamics of flow in the fluid region with that in the underlying porous medium, as described using the Darcy-Brinkman equations. The numerical model results are discussed and compared with previous analytical approximations. It is shown that the magnitude of the wave energy dissipation is determined by the depths and characteristics of the two fluid layers, with maximum energy decay occuring when the depth of the fluid mud layer is slightly greater than the viscous boundary layer formed at the mud interface. The wave-induced flow regimes in the boundary layers are studied and illustrated using vector plots of particle velocities in the boundary regions.

Explosive Forming of Stainless Steel Plate by Under Water Shock Wave

Y. Hidaka, H. Maehara, S. Itoh
Kumamoto University, Japan

When we process the difficult processing material in an existing metallic processing technology, though it depends on the size, cost and time hang in the production of plinth etc. Though it only has to mass-produce, for the special order, the cost hangs very much and it is non-economical. So we have tried to improve the formability limit of stainless steel plate and its method by improving the plinth made by stainless steel. The Explosive Forming is one of the metalworking methods. It has the advantage such as not receiving of the heat influence. In this method, the shock wave generated by the explosion and it through a suitable pressure medium like as a water or air act on a metal plate. In this research, we produce new plinth for explosive forming and it aimed at the improvement of the formability limit of stainless steel plate in various conditions of thickness of stainless steel plate. Moreover, after obtaining the amount of the transformation of the desire, the experiment is similarly conducted again, so we confirm the reproducibility.

Particle Filtration Processes in Deformable Media

M. Matalan, W. Brandstätter, G. Boiger, R. Bouwman
ICE-Strömungsforschung GmbH, Austria

In filtration processes it is necessary to consider both, the interaction of the fluid with the solid parts and the effect of particles carried in the fluid and accumulated on the solid. Both physical phenomena will be adressed in this paper. In the first part a closer look is taken to the interaction of the fluid with solid regions. Pressure and traction forces induced by the fluid motion on the solid domain, lead to certain deformations of the solid part. According to this multiphysical problem, it is important to couple the differential equations of fluid motion, namely the Navier Stokes equations and structural mechanical equations for the solid region, i.e. The Hooke's law. For their numerical discretization only one single computational mesh is used. This grid is changing with time and hence is recalculated at each time step to adjust to the deformation. This is important to conserve geometric consistency.
The second part of the presentation deals with simulation of large spherical particles. Those particles are injected into the fluid stream and cause deformation of the filter material. They are classified as “big particles”, which means that their size is much bigger than the size of a grid cell in the simulation grid. This is in contrast to simulation of “small particles”, where usually the temporal and spartial evolution of distribution functions via the solution of conservation equations of their respective moments is sought. The big particles contain a certain mass and momentum and interacts with the flow field. The main effects governing particle motion besides this are wall interactions, interactions with filter material and the collision between themselves and the plugging effect. The latter effect implies that with time, more and more particles are deposited on the solid walls. Thus, the effective solid surface is increased and this has an impact on the flow of the fluid. For simulation of these big particles a Discrete Phase Lagrange Model was developed. This model takes into account the two way coupling between the fluid and the particles. Those particles are related to Lagrange objects, which implies that their behaviour, such as their velocity, acceleration and position, is totally dependent on surrounding fluid flow conditions such as velocity and pressure. The simulation of big particles and the fluid structure interaction is realized in a single solver on the basis of the Open Source software OpenFoam [3]. The user can decide in an easy fashion, where particles should be injected and the degree of interaction between the fluid and the structure.

SESSION 3.4

MODELLING ADVANCES

FRIDAY 14 DECEMBER 2007
15:30 – 17:00

CHAIR

T Rahulan
University of Salford
UK

Modelling of Complex Process Conditions for the Calculation of Welding Residual Stresses and Distortion

M. Urner, E. Hanssen, T. Welters, K. Dilger
Technical University of Braunschweig, Germany

The manufacturing of welded structures excites stresses and distortions due to the inhomogeneous heat treatment. The integration of welding simulation into production planning offers the possibility to predict component properties after welding. The growing demands for reliable and flexible computational methods are generated by a decreasing average time to market and an intensified international competition. The welding distortion is the decisive factor concerning the production of welded components. For fatigue assessment under dynamic loading the knowledge of the overall stress state is essential because the additional stress component superimposes itself over the stresses resulting from external forces. Typical welding simulations are a combination of the computational description of different physical phenomena which are coupled with each other. At first an adequate derivation of the welding temperature field is required to predict the elastic-plastic deformation of components result to an inhomogeneous heat treatment. Furthermore a suitable reproduction of experimental knowledge about the elastic-plastic or viscoplastic behaviour at increased temperatures of the welded materials is essential, e.g. by means of appropriate constitutive equations. At this it has to be kept in mind, that especially steel materials may, in regions close to the weld seam, undergo phase transformations during the process and therefore may change their mechanical behaviour significantly. In most cases a suitable description of these transformations has to be achieved. The objective of the work is to present the techniques used nowadays for the welding simulation of arc welded structures, setting a special focus on the complex conditions occurring in welding processes with practical relevance. For instance mounting clamps and other heat sinks have a significant impact on the resulting temperature field in the welded component. Additionally the mechanical deformation of the component during the transient welding process is directly linked to the, often rigid but nevertheless finite, stiffness of the imposed restraints. Several spatiotemporal contiguous welding processes are often as much a challenge as the consideration of a realistic seam formation and phase transformations. The presented techniques are illustrated and reviewed with the help of different typical welding situations with practical relevance.

Modeling and Control of a Wheelchair for Mobility on Two Wheels

S. Ahmad
University of Sheffield, UK

This research is focusing on developing a wheelchair model and further on implementing suitable control strategies in order to lift the front wheels (casters) so that it achieves the upright position with the extension of height level. This research is aimed to help disabled people who are using the wheelchair as the main transport for mobility but cannot stand on his own due to permanent injuries on the extremities. It is important to note that a wheelchair on two wheels is expected not to take much space during mobility as compared to when it is on four wheels. Moreover, disabled people are encouraged and expected to perform most activities that others can do and hence lead an independent life. Therefore, this will then help them reach certain levels of height in confined spaces, e.g. to pick and place things on shelves. Furthermore, they will be able to participate in conversations at eye-to-eye level comfortably as normal people do that may facilitate psychological of disabled. In fact, researchers have accordingly shown a great deal of interest in advancing the current technology of wheelchair to let disabled people perform most of their daily life tasks independently. They have also proved that using wheelchair as a means of mobility is more efficient than walking. Thus, wheelchairs on two wheels are needed for disabled persons to perform some of the essential tasks in their living and work environments.
Wheelchair on two wheels is a great control issue to study due to its highly nonlinear and complex characteristics. The challenge in designing wheelchair on two wheels involves the design and implementation of suitable control strategies for a two wheeled wheelchair to perform comparably similar to a normal four wheeled wheelchair. In this research the State Space model of the standard wheelchair is first developed and tested by designing novel Linear Quadratic Regulator (LQR Control) for lifting up the chair (transforming a four-wheeled wheelchair to a two-wheeled wheelchair) and maintaining stability and balance while on two wheels. The results obtained demonstrate that the LQR approach works well on highly nonlinear systems such as the wheelchair system on two wheels and gives good system performance.

The Immersed Boundary-Lattice Boltzmann Method for the Unsteady Fluid-Particle Interaction

S. Farnoush – *Sharif,* M.T. Manzari, M.S. Seif
Sharif University of Technology, Tehran, Iran

The unsteady fluid particle interactions are numerically investigated by developing a suitable lattice Boltzmann method in the framework of immersed boundary scheme. Investigating unsteady fluid-particle interactions is a crucial step in the simulation of particulate fluid flows. For the first time, Feng & et al. (2004) used the idea of the immersed boundary schem introduced by Peskin (1972) in the lattice Boltzmann model, as a flow solver, to simulate particulate flows. The use of immersed boundary in embedding solid boundary via a source term in a lattice Boltzmann flow solver, removes the fluctuations of the forces and velocities arisen in the traditional lattice Boltzmann bounce-back rule for solid-fluid boundary condition. Afterwards, Feng & et al. enhanced the method by incorporating direct force approach of immersed boundary method (2005), which is called Proteus, instead of the penalty scheme of Feng's in 2004. Proteus is just capable of simulating steady incompressible flows. We have enhanced Proteus in two ways. First, we take the advantages of the idea of simulating unsteady incompressible flow with lattice Boltzmann method (He et al., 1997). Second, we improve the velocity projection in immersed boundary part of the Proteus. These two modifications enable us to investigate unsteady particulate flows.
In this paper, first, it is shown that this method is capable of simulating unsteady von-Karman Street behind a cylinder both qualitatively and quantitatively. The drag and lift coefficients are in a very good agreement with previous results (for Re=100). Then, the method is used to investigate the interaction of fluid with a fixed rigid particle in an unsteady Womersley flow.

Keywords: Lattice Boltzmann method, immersed boundary method, unsteady incompressible flow, Womersley flow, particle-fluid interaction

Fault Diagnosis of Rotating Machineries based on Gabor Order Analysis and Statistical Inference

F. Aminravan, F. Bakhtiari-Nejad
Amirkabir University of Technology, Iran

In this article rules for a fault diagnosis system based on Gabor order analysis and hypothesis testing for a transmission system working under run-up stage are proposed. The waveform-reconstruction schemes of order tracking such as the Gabor technique can extract specific order and spectral components by masking Gabor decomposed atoms in a Gabor lattice in addition to characterizing the processed signal in time-frequency domain. On the other hand, improved Gabor order analysis scheme is a combination of two families of joint time-frequency analysis and order analysis which overcomes order-crossing and resonance excitation problems that occur in other FFT based order analysis techniques.The study uses Gabor order tracking technique and an improved version of it to handle the order-crossing problem at resonance frequency occurrence, and then objectively these methods are used on a damaged gear-set for extracting order features which contain most information about faults. In this research, it is shown that differences between results obtained from original and modified Gabor atoms reconstructed orders could be considered as defect indicators. In compare with other conventional methods of order analysis like resampling based methods, this method has faster convergence and provides order time waveforms which give possibility of performing more statistical analysis. Implementation of proposed method on transmission system with various faults and classification of faults using statistical inference by means of differences obtained from two schemes show the effectiveness of proposed fault diagnosis algorithm based on extracted Gabor / modified Gabor orders as fault features.

Keywords: Joint time-frequency analysis; Gabor order analysis; Rotary machine monitoring/diagnostics; Hypothesis testing

Identification of Noise Emission of a Gear Unit

Aleš Belšak (1), Jurij Prezelj (2)
(1) University of Maribor, Faculty of Mechanical Engineering, Slovenia,
(2) University of Ljubljana, Faculty of Mechanical Engineering, Slovenia

Noise source visualization is an important tool in technical diagnostics. As a rule individual visualization methods are restricted to certain types of noise sources in a specific acoustic environment and in a limited frequency area. A visualization method of complex noise sources using an acoustic camera is dealt with. A new algorithm, which is applied, makes visualization of all types of different complex noise sources possible, which enables advanced determination of the condition of mechanical systems by means of noise. Faults and damages, which can cause problems associated with gear unit operation, are numerous, a fatigue crack in a tooth root being the least desirable of them. Its result is usually failure of gear unit operation. On the basis of numerical simulations of real operating conditions, real faults and damages have been produced in gear units. A laboratory test plant has been used to perform tests. Fault analyses based on this are presented. It is possible to define damage by monitoring noise. Impacts of various defects of a single-stage gear unit upon the vibrations that they produce are presented. Changes in tooth stiffness are produced by a fatigue crack. Changes of other dynamic parameters are more expressed when speaking of other faults. Time signals have been obtained by experiment and then analysed in relation to a hybrid procedure with the purpose to determine the level of non-stationarity of operating conditions, primarily, of rotational frequency. Signal analysis concerning a non-stationary signal has been used. Damages are associated with typical spectrogram patterns that indicate the presence of faults in a very reliable way.

Keywords: Gear, Failure, Noise Emmission, Signal Analysis, Acoustic Image

www.multiphysics.org