STAMS 2019
First Colloquium of the Spanish Theoretical and Applied Mechanics Society

PROGRAMME
STAMS 2019 Colloquium

FIRST COLLOQUIUM OF THE SPANISH THEORETICAL AND APPLIED MECHANICS SOCIETY

PROGRAMME

March 28 - 29, 2019
Madrid, Spain
Preface

The colloquium aims to bring together leading researchers in the areas of solid and fluid mechanics to engage in in-depth and far-ranging scientific discussion of current topics. Through this colloquium, we also wish to recognize and celebrate the life accomplishments of Enrique Alarcón and Amable Liñán and their pioneering roles in the development of Theoretical and Applied Mechanics in Spain.
Organizing institutions

STAMS2019
http://congreso.us.es/stams2019/

SEMTA
https://www.semta.org.es/
Colloquium Organization

Chairs

Pilar Ariza Moreno  Carlos Martínez Bazán
University of Sevilla  University of Jaén

Scientific Committee

Manuel Doblaré Castellano (University of Zaragoza)
Jaime Domínguez Abascal (University of Sevilla)
César Dopazo García (University of Zaragoza)
Javier Jiménez Sendín (Technical University of Madrid)

Local Organizing Committee

Pilar Ariza
Héctor Cifuentes
Alberto Fraile
Javier Ramos
José David Ríos
Colloquium Information

Conference Venue

The STAMS First Colloquium of Spanish Theoretical and Applied Mechanics Society place at Escuela Técnica Superior de Ingenieros Industriales de Madrid, José Guitiérrez Abascal, 2. 28006 Madrid. All talks are held at Salón de Actos (Thursday) and Sala C (Friday). The poster session and coffee breaks take place in the Sala de la Máquina (in the hall of the school).

Registration

On Thursday, from 8:15 AM to 9:00 AM. You may find the registration/information desk in the hall of the school. Upon your registration, you receive an identification badge. Please wear your identification badge visible throughout the colloquium since it serves as admission ticket for events of the colloquium.

Posters Session

Each author is responsible for mounting his/her material prior to the opening of the poster session and for removing it INMEDIATLY after the close of the session.

Poster numbers supplied by STAMS will be placed in the upper corner of each poster board. This number corresponds with the number assigned to each paper in the programme book.

It would be convenient that some of the authors stay with their poster during the poster session.
### Colloquium Programme

<table>
<thead>
<tr>
<th>Hour</th>
<th>Thursday</th>
<th>Friday</th>
</tr>
</thead>
<tbody>
<tr>
<td>8:15 - 9:00</td>
<td>Registration</td>
<td>9:30 - 10:15 Paul Clavín</td>
</tr>
<tr>
<td>9:00 - 9:15</td>
<td>Welcome</td>
<td>10:15 - 11:00 Juan Fernández de la Mora</td>
</tr>
<tr>
<td>9:15 - 9:45</td>
<td>Amable Liñán laudatio</td>
<td>11:00 - 11:40 Coffee</td>
</tr>
<tr>
<td>9:45 - 10:30</td>
<td>Juan C. Lasheras</td>
<td>11:40 - 12:25 Ramón Gilsanz</td>
</tr>
<tr>
<td>11:55 - 12:25</td>
<td>Enrique Alarcón laudatio</td>
<td>13:40 - 14:10 Enrique Alarcón</td>
</tr>
<tr>
<td>12:25 - 13:10</td>
<td>Michael Fardis</td>
<td>14:10 - 16:10 Lunch</td>
</tr>
<tr>
<td>13:10 - 13:55</td>
<td>Miguel Ortiz</td>
<td></td>
</tr>
<tr>
<td>13:55 - 16:00</td>
<td>Lunch</td>
<td></td>
</tr>
<tr>
<td>16:00 - 18:00</td>
<td>Poster session</td>
<td></td>
</tr>
<tr>
<td>18:00 - 18:30</td>
<td>Coffee</td>
<td></td>
</tr>
<tr>
<td>18:30</td>
<td>SEMTA general assembly</td>
<td></td>
</tr>
<tr>
<td>20:30</td>
<td>BANQUET</td>
<td></td>
</tr>
</tbody>
</table>
Sessions Programme: Thursday, March 28

<table>
<thead>
<tr>
<th>Session 1. Chair: Carlos Martínez Bazán.</th>
<th>(Salón de Actos)</th>
</tr>
</thead>
<tbody>
<tr>
<td>9:15 – 9:45</td>
<td>Amable Liñán laudatio</td>
</tr>
</tbody>
</table>
| 9:45 – 10:30 | SAVE - Strain Accumulation Venticulomegaly Hypothesis for the onset of idiopathic Normal Pressure Hydrocephalus (iNPH).  
Juan C. Lasheras  
University of California at San Diego |
| 10:30 – 11:15 | Flow and transport in the spinal canal.  
Antonio Sánchez  
University of California at San Diego |

<table>
<thead>
<tr>
<th>Session 2. Chair: Alfonso Fernández Canteli.</th>
<th>(Salón de Actos)</th>
</tr>
</thead>
<tbody>
<tr>
<td>11:55 – 12:25</td>
<td>Enrique Alarcón laudatio</td>
</tr>
</tbody>
</table>
Michael Fardis  
University of Patras |
Miguel Ortiz  
California Institute of Technology |
Sessions Programme: Friday, March 29

<table>
<thead>
<tr>
<th>Session 1. Chair: Pedro García Ybarra. (Sala C)</th>
</tr>
</thead>
</table>
Paul Clavin  
Aix-Marseille Université |
| 10:15 – 11:00 | Two high resolution fluid flow instruments for sizing nanoparticles: The evolution of the differential mobility analyzer, and the design of a hypothetical sizing condensation particle counter.  
Juan Fernández de la Mora  
Yale University |

<table>
<thead>
<tr>
<th>Session 2. Chair: Juan José Benito Muñoz. (Sala C)</th>
</tr>
</thead>
</table>
Ramón Gilsanz  
Gilsanz Murray Steficek LLP |
Ronaldo Borja  
Stanford University |
| 13:10 – 13:40 |  
Amable Liñán |
| 13:40 – 14:10 |  
Enrique Alarcón |
CARDIOVASCULAR diseases are the biggest source of health expenditure around the world [1]. An important subgroup is the one dedicated to the study of stenosis in coronary arteries. The stenoses are mainly caused by the growth of an atheromatous plaque that reduces the vessel lumen. This geometry change modifies the flow inside the arteries and can lead to a failure in the myocardial irrigation. To perform a functional evaluation of a coronary stenosis, the Fractional Flow Reserve (FFR) calculation has established itself as the gold standard beyond other procedures [2]. As getting this data requires an invasive treatment, several groups are chasing an accurate estimation applying Computational Fluid Dynamics (CFD) to medical images from a Coronary Computed Tomography Angiography (CCTA) [3,4]. The reliability of this estimation depends on the limitations of the geometry reconstructed from the images [5].

In this work, we evaluate the influence of the global threshold used in the image segmentation on the calculated FFR. We analyzed three sets of medical images corresponding to real cases treated for stenoses in coronary arteries. We selected the nominal value of the threshold, 200 HU, by fitting the CFD prediction for the FFR to the experimental value measured with catheterization in one of the cases. Variations of ±10 HU in the global thresholding were applied to that case and two additional sets of images, so that eight additional geometries were obtained. We performed CFD simulations and computed the FFR for each coronary tree. The results show that these variations in the threshold lead to a 4% mean deviation of the FFR with respect to that of the nominal value 200 HU. In all the cases, the variation +10 HU reduced the FFR value below the recommended limit for coronary revascularization (FFR=0.8).

References
We derive general expressions of the aerodynamic force and moment for an arbitrary set of pitching and heaving foils from the vortical impulse theory in the limit of linearized potential, two-dimensional flow. Of special interest for studying the propulsion of some aquatic and aerial animals and small unmanned vehicles is the case of two oscillating foils in an aligned tandem configuration, for which analytical explicit expressions for the lift, thrust and moment are obtained. In the limit of large separation distance, we recover the expressions of a single oscillating foil for the forewing, but the hindwing is always affected by the unsteady wake of the forewing. Relatively simple expressions are obtained in the cases of two heaving foils and of two pitching foils, for which the optimal conditions generating a maximum thrust force and a maximum propulsion efficiency are mapped in terms of the non-dimensional parameters, particularly separation distance and reduced frequency, and discussed physically in relation to the optimal results for a single oscillating foil. Finally, the present potential results are compared with available experimental and numerical data, with good agreement for small amplitude of the oscillations and sufficiently high Reynolds numbers.

Supported by the Ministerio de Economía y Competitividad of Spain Grant No. DPI2016-76151-C2-1R
POSTER 3

STATISTICAL APPROACH TO MECHANICS OF VISCOELASTIC SOLIDS. A NOVEL PARADIGM FOR MODELLING TIME-TEMPERATURE SUPERPOSITION PRINCIPLE


E-mail: alvarezvadrian@uniovi.es, enrique.castillo@unican.es

A wide variety of materials in science and engineering exhibits viscoelastic character, i.e. biomaterials, polymers, cementitious and woods. Multiple time dependence of external factors, such as temperature, pressure and humidity, set up a complex mechanical behavior to deal with. A worldwide effort has been devoted to develop mathematical models for including those external effects into the viscoelastic characterization, whereby temperature and pressure are the most relevant ones from a practical perspective. Current models are based on the master curve concept, supported by the time-temperature superposition principle (TTS), allowing the parametric family of temperature-dependent modulus curves along the time to be univocally represented by a single curve related to a reference value of the external effect. Despite of being extensively used in practical/real applications, some mathematical inconsistencies are identified in those models revealing lack of fulfilment of the inherent conditions to be satisfied by master curves. As a result, the solutions are influenced by the criteria and experience of the user.

The aim of this work is twofold. First, to analyze the mathematical conditions for master curves to be feasible (Figure 1, left). Second, to use statistical cumulative distribution functions to model the effect of temperature on the relaxation curves. The master curve is derived for a reference temperature based on the assumption of temperature acting as a scale-effect on the viscoelastic relaxation modulus curve. Finally, the applicability of this proposal is exemplified with relaxation tests results on polyvinyl-butyrall (PVB). (Figure 1, right).

Figure 1. Scheme of TTS principle (left) and example of application on PVB (right).
Modelling rubber-like materials has always been essential for some industrial applications, but it has received additional attention for its applicability in the modeling of biological tissues. Rubber-like materials are considered to be a network of polymeric chains that under deformation produce a non-linear stress-strain behaviour for the continuum. In hyperelasticity, the existence of a strain energy potential guarantees the exact fulfillment of the integrability conditions and that the material response is conservative. Traditionally the shape of the energy function is predefined and just dependent on some parameters that are determined using optimization techniques to fit experimental data points. That approach is followed in both the phenomenological approach and in the structure-based one. The main difference between the structure-based and phenomenological models is that structure-based models construct the macroscopic strain energy function from constituents’ behaviour while phenomenological models consider the material as a whole. Structure-based models are usually more complex but require fewer tests to give more reliable results under a general loading condition. For isotropic materials, the Ogden model (parameter-based, phenomenological) and the eight-chain model (parameter-based, structure-based) are common choices.

On the other hand, phenomenological WYPiWYG hyperelasticity follows a numerical approach to determine the macroscopic energy function. The energy function is determined solving the equations of the tests from which experimental data are available, with no need to redefine the shape of the energy function, and as a consequence, without employing optimization algorithms to determine any material parameter. Observed experimental behavior for independent tests is captured to any desired accuracy. In this work, a Micro-structural nonlinear elastic model based on WYPiWYG hyperelasticity is presented. Since it is a structure-based model it just requires a single test to characterize the constituents’ behaviour. Additionally, the predictions obtained for a general case improve significantly compared to other phenomenological and structure-based models. To determine the behaviour of the material under any deformation, first the fiber/chain stress-strain response is determined from macroscopic test/s. Then assuming an isotropic distribution of fiber/chains, the continuum behaviour is reproduced for any arbitrary condition. To provide this information in an efficient way for general finite element simulations, constitutive manifolds can be constructed.
Contrary to birds, bats and most insects, dragonflies have two pairs of wings arranged in a tandem configuration which can actuate independently. By modifying the kinematics and the phase relationship between the fore and hind pair of wings, dragonflies are capable of performing rapid manoeuvres and cruising efficiently. Although several studies have analysed the kinematics of tandem flight that provides an optimal configuration in terms of propulsive efficiency and/or thrust, they are usually restricted to two dimensional (2D) configurations. Consequently, the applicability of such optimal kinematics is not straightforward in a three-dimensional (3D) configuration, since the wing shape and actual motion of the wing (dragonflies’ wings do not heave but flap about an axis) are not defined in a 2D configuration. Therefore, the aim of this work is to investigate the changes in the efficiency when a 2D optimal kinematics is used in wings of finite span. In particular, we consider two flat, identical wings, with a rectangular planform and constant thickness. Two different planforms, $AR = b/c = [2, 4]$, are considered, where $c$ is the chord of the wing and $b$ is the wingspan. The motion of the wing is based on a 2D optimal kinematics which combines heaving and pitching about the mid chord. We consider two kinds of motion for the wings: heaving (all chordwise sections of a wing move with the same velocity); and flapping (i.e., rotation of the wing about a fixed axis) about an axis parallel to the free-stream. In this way, the effect of the aspect ratio and the effect of flapping instead of heaving can be analysed. The results show that the interaction between the vortical structures shed by both wings is similar for the 2D kinematics and for the 3D heaving wings; thus, the resulting propulsive efficiency is similar. Nevertheless, the aerodynamic forces are smaller for lower $AR$. When the heaving and flapping motions are compared, the vortical structures are qualitatively similar; however, when the wings are flapping, smaller aerodynamics forces are produced and a detriment in the propulsive efficiency is obtained.
A MATHEMATICAL AND COMPUTATIONAL MODEL FOR THE SIMULATION OF INTERACTING CELLULAR POPULATIONS IN MECHANICAL ENVIRONMENTS. APPLICATION TO GLIOBLASTOMA CELL CULTURES IN MICROFLUIDIC DEVICES

J. Ayensa-Jiménez*, M. H. Doweidar**, M. Doblaré*

*Aragon Institute of Engineering Research, University of Zaragoza, C/ Mariano Esquillor s/n, 50018 Zaragoza, Spain.

**School of Engineering and Architecture (EINA), University of Zaragoza, C/ Maria de Luna s/n, Edificio Betancourt, 50018 Zaragoza, Spain.

E-mail: jacoboa@unizar.es, jacoboa@unizar.es, mdoablare@unizar.es

Multi phenotype cellular microenvironments are extremely challenging biological systems to be modeled. In such a case, many factors should be taken into account such as chemical interactions with nutrients or other biochemical signals, physical driving factors such as electrical or thermal cues, cell proliferation, migration, differentiation, and apoptosis. Besides, the two-way coupling between the alive cells and the inert mechanical matrix (mechanotaxis and tissue remodelation) add more difficulty, which makes the resolution of the mechanical problem mandatory, up to convenient simplification hypotheses.

Mathematical models are powerful tools to understand the complexity of these events and their interactions. They offer a cheap and flexible way to explore therapeutic strategies or drug testing in clinical diseases such as cancer. We present a continuum-based mathematical approach to the interacting cellular population’s problem that takes into account the mechanical evolution and matrix remodelling, particularized for viscoelastic mediums. The governing equations are the transport equations for cell populations and chemical species as well as the conservation laws (linear and angular momentum) for the culture medium, besides to a viscoelasticity model relating stresses and strains. In addition, chemotaxis, mechanotaxis, proliferation, differentiation and apoptosis terms, considered in the balance, are all expressed as a function of field variables, resulting in a nonlinear framework. Finally, the evolution of the mechanical parameters depend of cell populations according to the appropriate remodeling rules, enabling backward feedback. The methodology is illustrated in the study of the formation of migrating cell structures, known as pseudopalisades, appearing around necrotic cores in Glioblastoma Multiforme, the most destructive primary brain cancer. The availability of data from experiments, carried on in microfluidic devices, allows model characterization, simplification and parameter tuning, which is in line with our in silico results.

The presented computational model proves to be a helpful tool for the set-up of new complex biological models that include cell-substrate evolution and interactions. More experimental campaigns are needed to define and calibrate these models.
POSTER 7
FRACTURE CHARACTERIZATION OF EPOXY ADHESIVES USING PRE-CRACKED CT SAMPLES

M. Casasola*, M. J. Lamela*, A. Fernández-Canteli*, Miguel Muñiz-Calvente*,
F. Pelayo*, Adrián Álvarez-Vázquez*, A. Salazar**, J.M. Pintado***

*University of Oviedo, 33203 Gijón, Spain.
**University Rey Juan Carlos, 28933 Móstoles, Spain.
***Instituto Nacional Técnica Aeroespacial, 28850 Torrejón de Ardoz, Spain.
E-mail: pintadojm@inta.es, alicia.salazar@urjc.es, mjesuslr@uniovi.es

The continuous search for new materials that provide equivalent or superior stiffness and strength to those currently being used is the principal goal of structural components design, particularly, in aeronautics. In this paper the critical stress intensity factor ($K_c$) and the critical strain energy release rate ($G_c$) are determined for the epoxy resin EPOLAM 2025 following the D5045-99 standard and using Compact Tension (CT) specimens. This resin is mainly utilized as an adhesive joint and as the matrix in fiber reinforced plastics.

Given the difficulty to introduce a pre-crack at the tip of the notch of the CT sample that fulfils the standard requirements, alternative experimental methods are attempted. Furthermore, a finite element analysis using the commercial Abaqus code is carried out to obtain the fracture parameters numerically and to study the relationship between crack length and $K_i$ (Figure 1). For that, CT specimen models in 2D and 3D are used, in which the pre-crack was defined with a contour integral (Figure 2).

The fracture parameters obtained experimentally are within the range of values expected for such a material, despite not being precise because of the inaccuracy of the pre-cracks. No alternative pre-cracking method was found to be more adequate than the one performed in this work, but its efficiency is proved numerically by analyzing the stress intensity factor and the Von Mises stress distribution for different loading types, which allowed a visualization of the plastic zone size at the tip of the pre-crack.

Figure 1. Evolution of $K_i$ for 3D and 2D CT models. Figure 2. 3D CT specimen model.
Supersonic ejectors are used in a wide range of applications such as compression of refrigerants in cooling systems, pumping of volatile fluids, or vacuum generation. In this latter case, also known as zero-secondary flow, physics in the vacuum ejector is more complicated than in steady cases, since recirculation bubbles existing in the diffuser exhibit transient behavior during the start-up period. This numerical study has been performed with open source CFD toolbox OpenFoam v.6 in a 64 AMD Opteron cores cluster. It is well known that supersonic flow is better simulated with density-based solvers. The main inconvenient of OpeFoam transient density-based solvers is that they are explicit, based on Kurganov and Tadmor central-upwind schemes, and, hence, time step has to be very small in order to keep simulation stable. Also, it has been reported that implicit solvers are much efficient and accurate that explicit solvers. For the present work the open access implicit density-based solver HiSA[1], that implements the AUSM+up upwind scheme for face fluxes, has been used. This solver allows to solve unsteady flows with large Courant numbers than explicit solvers. Numerical results with the HiSA solver using parallel simulation are presented. Ejector performance obtained with the simulation is compared with experimental results from laboratory tests.

Keywords: zero-secondary flow ejector, vacuum supersonic ejectors, shock waves, density-based compressible solvers

References

Aerosolized nanoparticles can be used as building blocks for materials synthesis leading to coatings and thin porous layers, which retain a large surface/volume ratio as distinctive feature. Aerosol deposits acquire a structure controlled by the form of the constitutive particles and by the way in which these particles arrive and attach to the forming material (Castillo et al., 2014, Rodriguez-Perez et al., 2005, 2007). These nanostructured porous coatings have many applications, in particular those requiring materials with a large active surface, as is the case of catalytic layers. Nanostructured materials are prepared from the steady electrohydrodynamic atomization of catalytic inks (Castillo et al., 2018, Martin et al., 2012) and tested as fuel cell electrodes (Martin et al., 2017, 2018, Martinez-Vazquez et al., 2015). Adjusting the electrospray control parameters (needle voltage, collecting plate voltage and ink flow rate), a steady cone-jet electrospray is maintained leading to well-structured and regular catalytic layers. The electrodes show extremely large specific power (reaching a power of 21 kW per gram of Pt in the cell) due to the properties of the layer: large porosity, uniform distribution of the catalyst and high density of (Pt) active sites.

Work supported by Ministerio Español de Economía, Industria y Competitividad, Ministerio de Ciencia, Innovación y Universidades, and partially supported by the European Regional Development Funds (FEDER) under projects ENE2015–67635-R, RTC-2017-6730-3, and RTI2018-099143-B-I00.

References


In a wide range of applications involving chemically reacting flows, thermochemical calculations involving overall mass and energy balances with chemical equilibrium are enough to estimate the overall performance of the system. This is often determined by magnitudes such as the adiabatic flame temperature or the equilibrium composition of the products mixture. Thermochemical codes have been developed for this purpose starting in the late 1940’s at the NACA Lewis Flight Propulsion Laboratory, with the initial aim of evaluating rocket performance of various potential propellants. The use of thermochemical codes has been growing ever since, and today they are basic research tools both for combustion applications and in the analysis of the thermal decomposition of high-energy materials (HEMs), including both deflagration and detonation conditions. As a first step towards the development of a wider-scope thermochemical tool, in this work we present a thermochemical code with application to gaseous combustion problems recently implemented by the authors in MATLAB®. The code solves six chemical equilibrium problems (TP, HP, SP, TV, EV and SV transformations; where T denotes temperature, P pressure, H enthalpy, S entropy, E internal energy and V volume), incident and reflected planar shock waves, as well as ideal detonations according to Chapman-Jouguet theory, assuming always ideal gases. The code computes the equilibrium composition using equilibrium constants rather than by minimization of the Gibbs–Helmholtz free energy and employs NASA’s 9-coefficient polynomial fits to evaluate the thermodynamic properties. Besides the plain code, the new tool has been equipped with a Graphical User Interface (hereafter Combustion-Toolbox) developed in MATLAB® 2018b under AppDesigner. The results obtained with Combustion-Toolbox have been validated against, and are in good agreement with, NASA’s Chemical Equilibrium with Applications (CEA) program, CANTERA, and Caltech’s Shock and Detonation Toolbox. Moreover, the time required for the computations is comparable to that of other existing codes. Combustion-Toolbox has teaching and research aspirations and will be distributed as open source package as soon it has been fully tested.
We postulate a general formulation for the Eulerian-Eulerian conservation equations governing the multispecies and multiphase gas-solid flows with mass transfer and chemical reaction. The model, based on the Kinetic Theory of Granular Flows, has been implemented in the open-source platform OpenFOAM.

A pilot-scale gasifier fluidized bed with limestone and coal is simulated. With the aim of providing some insight on the influence of article polydispersion on flow patterns and species distributions, we compare solutions obtained by modeling the bed with a single or with seven coal-particle sizes, and also as a reactive or inert bed.

Results reveal that the preferential motion of the smaller coal particles towards the bed top significantly delays char consumption at the bottom layers, favors volatile oxidation, and results in a more uneven lateral distribution of the off-gas.

Figure 1: Volume fraction for limestone (most left) and coal phases (ordered from the largest (s1) and smallest (s7) diameter, from left to right) in the reactive bed
EXPERIMENTAL STUDY ON HIGHLY DEFORMABLE COUPLING BEAMS


*Department of Mechanical Engineering, Polytechnic University of Madrid, José Gutiérrez Abascal, 2 28006 Madrid, Spain.

**Department of Civil and Structural Engineering, The University of Sheffield, Sir Frederick Mappin Building, Mappin Street, Sheffield, S1 3JD (UK)

E-mail: d.escolanod@upm.es, samarraffoul@hotmail.com, m.guadagnini@sheffield.ac.uk, k.pilakoutas@sheffield.ac.uk

Reinforced Concrete (RC) structural coupled walls have long been used in medium to high rise buildings as a primary lateral force resisting system for wind and seismic loads. When a wall system is subjected to large lateral actions, the coupling beams are required to transfer high shear forces and undergo large deformations. Coupling beams are called to dissipate energy over the entire height of the wall by providing substantial deformation ductility. Therefore, the performance of the entire lateral resisting system depends on the deformation capacity of the coupling beams.

This poster presents a novel solution for highly deformable coupling beams using confined Highly Deformable Concrete and presents the results of a preliminary experimental investigation. Highly Deformable Concrete utilises recycled rubber particles as replacement for both fines and coarse aggregates. Although the inclusion of rubber in concrete can lead to a significant reduction in compressive strength and stiffness, rubberised concrete exhibits higher deformability, ductility and energy dissipation capacity than traditional concrete when confined with steel jackets or FRP.

The results of an experimental program on 1:2 scale coupling beams with regular and confined Highly Deformable Concrete are discussed to assess the performance of the proposed novel solution. Two different reinforcement details (flexural and diagonal) are considered. All specimens are subjected to the same loading history comprising a series of quasi-static load cycles at increasing levels of maximum displacement. The results of the tests indicate a better seismic performance of the confined Highly Deformable Concrete coupling beams, in terms of ductility, deformability and energy dissipation capacity compared to conventional RC coupling beams.

Keywords: Rubberised Concrete, Coupling beams, Confinement.
Background: The WHO estimates that in 2017 approximately 15 million babies will be born preterm, this is, a rate above 1 in 10 newborns [1]. Worldwide, complications of preterm births have supplanted pneumonia as the primary cause of child mortality [1,2]. While progress is being made in identifying socioeconomic risk factors of preterm birth, the biology of cervical ripening that leads to birth is poorly understood. Currently, there is no clinical tool to quantitatively evaluate the cervical biomechanical state. Ultrasonic characterization and understanding of soft tissue has been developed as a clinical diagnostic tool over the last two decades [3] and evolved through different technologies: quasi-static, dynamic elastography, based acoustic radiation force: ARFI, vibroacoustography or pSWE, or on direct excitation: sonoelelastography and our emerging torsional wave principle [4].

Aims: We work on enabling new sensor technologies linked to soft tissue biomechanics, to endow a new class of biomarkers that quantify the mechanical functionality of the cervix, and indeed any soft tissue. Beyond labor disorders, abnormalities in the structural architecture of soft tissues are intimately linked to a broad range of pathologies including tumors, atherosclerosis, liver fibrosis etc. The unexplored nature and applicability span of mechanical biomarkers and torsional waves endows a foundational diagnostic technology.

Methods: Existing ultrasonic techniques are restricted to map first order tissue stiffness. In contrast, our recent advances covering (a) torsional waves (shear elastic waves that propagate in quasifluids radially and in-depth in a curled geometry), (b) sensors (based on a novel arrangement of concentric sandwiches of piezo- and electro-mechanical elements), (c) propagation models and (d) patient testing, are allowing to quantify the mechanical functionality through relevant parameters beyond linear: dispersive and nonlinear.

Results and Conclusions:
- To understand how the structural architecture of soft tissue is intimately linked and controls a broad range of pathologies, which underpins the foundation of new diagnostic technology.
- To develop new sensor technologies capable of effectively sensing tissue elasticity and yield simple and robust diagnostic tests and instruments.
- To ground a new generation of biomarkers of physical nature based on the mechanical micro architecture and properties of the tissue.

References

Carbon Fiber Reinforced Polymers (CFRP) materials lays in the category of “difficult to cut” materials because of the hard and abrasive character of the fibers embedded in the polymer matrix. The use of this material in aerospace industry is common and frequently manufactured close to the final shape. However, drilling operations are needed to join different parts of huge structures. The damage induced during drilling is the origin of significant number of rejected pieces.

With the objective to reduce the damage during drilling, the use of numerical models validated with experimental results is of crucial importance, giving an alternative to costly and complex experiments. Despite the interest of simulating composite machining, only few works in technical literature deal with finite elements analysis of these processes. Most of the works referred in literature simplify the process considering the drill bit as a punch with the objective of reducing the computational time. Even though this advantage, the main problem of this type of models is the impossibility of simulate chip removal.

In this work a 3D numerical model based on the use of finite elements has been developed. The geometry of the drill bit and the workpiece were reproduced and the cutting and feed movements were imposed in order to simulate chip removal and analyze the influence of different factors in delamination extension. The model reproduces not only the inter-laminar damage with cohesive elements as well as the intra-laminar damage using a VUMAT subroutine implemented in ABAQUS/Explicit.

The model has been validated in terms of thrust force and exit delamination prediction for different values of feed rate and cutting speed. Experimental data were obtained from previous work published by the authors.

Funding: Financial support for this work has been provided by the Ministry of Economy and Competiveness of Spain in the project DPI2017-89197-C2-1-R y DPI2017-89197-C2-2-R
This work focuses on the study of free rising motion and deformation of bubbles in still liquids in the presence of walls. The relevant dimensionless parameters of the problem are the Bond, \( Bo = \frac{\rho g D^2}{\sigma} \) and Galilei, \( Ga = \frac{\sqrt{g D^3}}{\mu} \) numbers, density and viscosity ratios, and the distance to the wall, \( L/D \).

For this purpose, the Open Source Software Basilisk (http://basilisk.fr/) has been used to solve the incompressible, variable-density Navier-Stokes equations plus a source term of surface tension coupled to a Volume of Fluid technique for the interface tracking. Three different cases have been analyzed, describing rectilinear, planar zigzag and spiral bubble paths respectively in the absence of walls. When a wall is placed near the bubbles, intriguing effects have been observed with striking changes in their trajectory and forces acting on their surface. For example, for a rectilinear case of \( Bo=5 \) and \( Ga=59.61 \), we have observed that the bubble rather than rising vertically, it hits the wall and bounces off as it rises up.

This study pursues the understanding of the bubble motion, accounting for the wall effect, determining the forces acting on the bubble. There are several papers studying bubble linear shear flows, but only a few of them focus on free rising bubbles. The main objectives are:

- Characterization of free rising bubble motion: path, wake and shape.
- Identification of different bubble trajectories, i.e. rectilinear, zigzag or spiral paths, as well as different wakes
- Influence of walls on the bubble motion: kinematics and dynamics.
- Measurement of bubble forces for different free rising paths.
- Performance of experimental and DNS analysis to compare results.

Nowadays, simulations of free rising bubbles will help to determine the bubble forces and the effect of the presence of walls. Some results of the present study are the following:

- \( Bo \) and \( Ga \) numbers determine the motion of free rising bubbles.
- Axisymmetric simulations can be used for rectilinear path and axisymmetric wakes.
- Knowledge of the forces allows to understand the bubble motion.
- Wall influence changes bubble motion.
- The bubble drag and lift forces are modified by the presence of the wall.

This work has been supported by the Spanish MINECO and European Funds under Projects DPI2014-59292-C3-3, DPI2015-71901-REDT, and DPI2017-88201-C3-2-R.
POSTER 16

EXPERIMENTAL EVALUATION OF THE ENERGY DISSIPATION CAPACITY OF RC COLUMNS FROM SHAKE TABLE TESTS OF STRUCTURES SUBJECTED TO BIDIRECTIONAL SEISMIC LOADING


1Department of Mechanical Engineering, Universidad Politécnica de Madrid, Jose Gutierrez Abascal 2-28006 Madrid, Spain.

2Department of Mechanical and Mining Engineering, University of Jaén, Campus Científico Tecnológico de Linares, Spain.

E-mail: david.gale@upm.es, jdonaire@ujaen.es, d.escolano@upm.es, guillermo.gonzalez.sanz@upm.es, amadeo.benavent@upm.es

The assessment of the structural performance under different scenarios of seismic hazard requires quantification of damage not only in terms of maximum deformation but also—and above all—in terms of dissipated energy (cumulative damage). Moreover, the most rational way to evaluate safety against earthquakes is by comparing the energy dissipation demand with the energy dissipation capacity of the structure. This is the basis of the seismic design approach based on the energy balance. One of the largest gaps of knowledge for the application of the energy-based approach to conventional structures concerns the evaluation of the energy dissipation capacity of structural elements and systems. Addressing this issue calls for tests on structural elements and structures subjected to realistic seismic loadings. The optimal and more realistic source of information for evaluating the energy dissipation capacity of structural elements is the shake table test conducted on full or partial structural models. This type of experiments are very costly, time consuming and the analysis of the results is cumbersome since all response quantities cannot be directly measured during the experiment. Therefore, it is often necessary to complete the test information with assumptions regarding constitutive laws for materials, levels of inherent damping, and numerical simulations. This study analyses the results of shake table test conducted by the authors on RC waffle flat plate structures supported on isolated columns under bidirectional seismic loadings. It shows how the energy dissipated at column ends can be estimated from the reading of the strain gauges attached to the steel, and the dependence of the results on the constitutive law assumed for the concrete.
TROCHOIDAL-GEAR PUMP TECHNOLOGY


*Department of Fluid Mechanics, LABSON & IAFARG, Universitat Politècnica de Catalunya, Colom 11 – 08222 Terrassa, Spain.

**Department of Mechanical Engineering, LABSON & IAFARG, Universitat Politècnica de Catalunya, Colom 11 – 08222 Terrassa, Spain.

E-mail: pedro.javier.gamez@upc.edu; esteban.codina@upc.edu; robert.castilla@upc.edu; javier.freire@upc.edu; carlos.rio-cano@upc.edu

Trochoidal-gear technology has evolved exponentially and is emerged as one of the leading research technologies in the last decade. Its benefits have an important number of sectors of implementation such as pharmacy (bottling of medicines) and medical (silicone application on medical disposables) in life science and coating applications, filling and dosing technology (additives, adhesives) and dispensing technology (printing inks), among others, in industrial, aeronautical and mechanical engineering.

The literature survey evinces the scattering between academia publications and improved products by the manufacturers. The academia research and results are hardly reflected in the manufactured products in the fluid power industry, especially regarding to gerotor pumps. Current commercial solutions of this technology are limited and typically tailor-made by the end-users. A well-known example is that the improvement in engine fuel economy has become one of the most global important objectives along with the growing mainstream popularity of hybrid vehicles. Since the engines used operate for shorter periods than conventional ones, the engine oil temperature remains lower and oil pressure increases due to its higher viscosity. Then, automakers, as end-users, in order to help resolve this issue, develop new concepts in oil pumps, fully variable and friction reduction. Another example, it is the extremely low values of manufacturing tolerances and surface finishing that fluid power products are requested to accomplish the everyday most demanding performance.

The work of our group intends to collect recommendations that combine academia and industry expertise to make better use of these extensive studies in this area. Three main methodologies concern these recommendations: prototypes with experimental research, numerical approaches and fault detection work. The advanced products by means of prototype research progresses to leakage-free and noiseless hydraulic systems, miniaturization, compact and new materials, magnetic-drive, greener and energy saving fluidic products.

Keywords: gerotor pump, best practices, diagnosis, modelling, efficiency, fluid power
**POSTER 18**

**EXPERIMENTAL AND CFD STUDIES IN STENTED ARTERIES AND IN THE LEFT ATRIAL APPENDAGE**

Javier García 1*, Alberto Pozo-Álvarez 2, Jorge Muñoz Paniagua 1, Francisco Castro 2, José Sierra-Pallares 2

1Departamento de Ingeniería Energética. Escuela Técnica Superior de Ingenieros Industriales. Universidad Politécnica de Madrid (Spain)

2Grupo de Ingeniería de Fluidos. Departamento de Ingeniería Energética y Fluidomecánica. Universidad de Valladolid (Spain)

E-mail: javier.garciag@upm.es

Our recent work resulting from the collaboration between the Fluid Mechanics Research Groups belonging to the University of Valladolid (UVa) and the Polytechnic University of Madrid (UPM) regarding blood flow in arteries and cardiac flow may be summarized as follows:

- Experimental and CFD study of blood flow in stented coronary bifurcations. The first objective is to validate a numerical model by comparison against experiments for steady and unsteady cases. A second objective is to compare the hemodynamic performance of one of bifurcation-specific new devices (Stentys) to that of conventional devices and stenting techniques in a simplified coronary bifurcation model. Different coronary bifurcation stenting techniques are analyzed focusing on factors contributing to restenosis, such as wall shear stress (WSS), oscillatory shear index (OSI), pressure loss, and local normalized helicity (LNH).

- Numerical study of blood clots influence on the flow pattern and platelet activation on a stented bifurcation model. Stent implantation can lead to other stenting-related problems. One of these is the deposition and accumulation of blood clots over stent struts. Numerical models are developed to take into account the effect on the flow pattern and platelet activation of blood clot depositions on the stent struts of stented coronary bifurcations. The numerical models are first validated with experimental measurements performed for this purpose, considering flows with suspended artificial thrombi, which naturally deposited on stent struts.

- Numerical study of bioreabsorbable stent degradation. One of the main uncertainties in the use of the new magnesium-coated stents is their behavior when deployed on coronary bifurcations. Their struts are wider and potentially may cause major hemodynamic flow disturbances. Additionally, their degradation mechanism is not only produced by the wall artery but also by the blood flow. A numerical model is presented that takes into account the flow pattern changes as stent degrades.

- In vitro and in silico study of flow patterns in the left atrial appendage (LAA). It is well known that atrial fibrillation (AF) is the most common cause of cardioembolic stroke. AF is associated with increased risk of stroke due to development of thrombi. Physiologic factors such as decreased atrial blood flow and increased stasis (left atrium (LA) velocities < 0.2m/s) are associated with thrombus formation in the LAA. The main objective is to develop a numerical tool capable of predicting the risk in relation to fluid dynamics parameters. To carry out this study an idealized LA and LAA are tested experimentally and the results are used to validate the numerical model.
We perform numerical simulations to investigate the fluid-structure interaction in the wake behind a D-shaped body with a rear cavity of flexible plates, focusing on the analysis of aerodynamic forces and plates’ flowinduced vibrations. The effect of the stiffness parameter of plates is studied for a given laminar Reynolds number to evaluate its role on the vortex shedding frequency, drag and lift coefficients and plates dynamics. A strong, implicit, partitioned approach has been used to solve the fluid-structure coupling, while the solid elastic behavior has been modeled following a nonlinear St.Venant-Kirchhoff law. The model has been first validated through comparison with a computational benchmark data available for a flexible splitter plate behind a cylinder, both in terms of frequency and plate displacement. The results show two different regimes in terms of solid and wake interaction, depending on the stiffness parameter. More precisely, for low stiffness values when the natural vibration of the solid is similar to the vortex shedding frequency, the plates oscillate featuring large amplitudes and vibrate at their natural frequency. Conversely, at large values of such stiffness parameter, oscillations are less energetic and the dynamics is coupled with that of the wake in terms of frequency. Moreover, symmetric and antisymmetric plates’ oscillations are also observed. The study also shows the potential of flexible plates for energy harvesting and flow control, in terms of drag reduction and vortex shedding amplitude and how different positions of the plates at the rear cavity affect them.
The local propagation velocity of stretched (curved & strained) premixed flames is known to depend linearly on the stretch when it is small; i.e., when the characteristic scale $\Lambda$ of the flame front deformation is large compared to the thermal flame thickness $\delta_\varphi$ such that $\varepsilon \equiv \delta_\varphi/\Lambda \ll 1$. The result, restricted to curvature, was first proposed by Markstein and later analytically derived by means of a multiscale asymptotic method in the limit $\varepsilon \to 0$ [1]. The validity of this result or extensions thereof for moderate values of $\varepsilon$ is an open question today because the intricacy of the required computations has prevented extension of the asymptotic analysis to higher orders in $\varepsilon$.

As an alternative to asymptotic analysis, we have used an experimental and numerical approach to try to ascertain the character of higher order corrections. Laminar jet flames of methane-air mixtures have been studied. A PIV system with olive oil droplets as flow tracers has been used to measure the gas velocity field upstream of the flame. Flame images also allowed to determine both the flame front position and its curvature along the front. A numerical code has been also developed for this type of jet flames and used for the same purpose.

A new regime of elongated flames displaying a faster-than-linear variation of propagation velocity with flame stretch has been found for large inlet gas velocities. The tip of these elongated flames is a hemispherical cap that moves faster than the flame tip in the Markstein regime, making the flame comparatively shorter. The transition between both regimes occurs suddenly at a critical value of the gas inlet velocity [2-4].

Work supported by Ministerio Español de Economía, Industria y Competitividad, Ministerio de Ciencia, Innovación y Universidades, and partially supported by the European Regional Development Funds (FEDER) under projects ENE2015-67635-R, RTC-2017-6730-3, and RTI2018-099143-B-I00.

References

The standard D'yakov and Kontorovich (DK) instability refers to planar shock waves that, once perturbed, oscillate with constant amplitude in the long-time regime. As a direct result, pressure perturbations generated right behind the shock propagate downstream as non-evanescent sound waves, an effect known as Spontaneous Acoustic Emission (SAE). For the DK-regime to be achieved, the slope of the Rankine-Hugoniot curve in the post-shock state must meet certain conditions, which have been usually related to non-ideal equations of state. This work reports that DK-instability, or SAE, can also occur in shocks moving in perfect gases when exothermic effects take place. In particular, a planar detonation, initially perturbed with a wavelength much larger than the detonation thickness, may exhibit constant-amplitude oscillations when the amount of heat release is positively correlated with the shock strength. The opposite highly-damped oscillating regime may occur when there exists a negative correlation with the shock strength and the heat release change. This work employs a linear perturbation model to describe both long-time and transient evolutions of the planar reactive front, assumed infinitely thin, as well as the sound and the entropy-vorticity field downstream.
POSTER 22
CRACK PATH PREDICTION UNDER FATIGUE BIAXIAL OUT-OF-PHASE LOADS IN CRUCIFORM SPECIMEN USING XFEMC
D. Infante-García*, H. Miguélez*, E. Giner**

* Department of Mechanical Engineering. Universidad Carlos III de Madrid, Avda. de la Universidad 30, 28911 Leganés, Madrid, Spain.

** Centre of Research in Mechanical Engineering – CIIM, Department of Mechanical and Materials Engineering, Universitat Politècnica de València. Camino de Vera, 46022 Valencia, Spain.

E-mail: eginerm@mcm.upv.es, dinfante@ing.uc3m.es

In this work, a crack growth numerical analysis is performed for the study of crack paths growing from an initial central crack in a cruciform specimen under biaxial fatigue loads. The crack propagation has been studied using linear elastic fracture mechanics and the extended finite element method (XFEM). The objective is to analyse the effect of different phase angles of loading ($\delta_{xy}$) and different angle inclinations ($\theta$) of the initial crack on the crack path. In addition, predictions are carried out using different orientation criteria for nonproportional loading available in the literature. On the other hand, stress intensity factor ranges in mode I and mode II of a crack either aligned or inclined to the load directions have also been analysed. Fig. 1 shows crack paths predicted for different initial central cracks under out-of-phase loading 90° using the criterion of max(\(\Delta k^*\)).

Figure 1: Crack paths predicted with the criterion of max(\(\Delta k^*\)) with phase angle of loading equal to 90° (thick black lines symbolize the initial cracks).

Symmetrical branching is predicted for an initial crack inclined at 45° with phase angle of loading equal to 90° and 180°, whereas co-planar crack growth is predicted either when a crack is aligned to the loading axes or under in-phase loading. Numerical results are in good agreement with the experimental observations found in the literature. The study reveals important differences in the crack path predictions depending on the orientation criteria.
The present work will introduce a method called higher order dynamic mode decomposition (HODMD) [1], which is an extension of the well-known technique dynamic mode decomposition (DMD) [2]. This extension has been introduced by the authors to detect spatio-temporal patterns in complex non-linear dynamics. The method combines singular value decomposition (SVD) with DMD [2] and Takens’ delay embedding theorem [3] to approximate the main dynamics describing a signal. The good performance of this method has been tested into a wide range of problems covering from non-linear dynamical systems (i.e.: complex Ginzburg-Landau equation, Lorenz system, …), to complex fluid dynamic problems [4] (i.e.: analysis of noisy experiments and transitional flows). Finally, the method called spatio temporal Koopman decomposition (STKD) [5] will also be presented, which is introduces the spatio-temporal HODMD analysis. This method is suitable to calculate flow structures, defined as traveling waves. Some examples include the thermal convection in a rotating spherical shell or the spatio-temporal flow structures describing the wake of a wind turbine.

References

The increasing production of greenhouse gases in the transport industry motivates the invention and design of strategies aiming at reducing the fuel consumption of different types of road vehicles. Aerodynamic forces, i.e. drag force, have considerable significance in relation with CO\textsubscript{2} emissions, as a result, numerous research studies are conducted to understand, characterize and control the flow around simplified models, similar to real vehicles, in order to reduce their environmental impact. This type of flow is characterized by the formation of a 3D wake behind the body and the shedding of complex vortices, which are strongly linked to drag force and fuel consumption. The present work investigates the use of improved passive control devices, in particular, rear cavity systems are chosen due to its simplicity and high efficiency as drag reducers.

The purpose of the present study is to design an optimal rear cavity capable to enhance the performance of classical drag reduction devices, as rear cavities are, employing gradient-based drag sensitivity by means of adjoint method.

Two different approaches, based on the inclusion of local force perturbations, are employed to optimize the effectiveness of rear cavities. The former consist in placing control cylinders in highest sensitivity regions where their influence in terms of drag reduction is greater. The latter introduce small deformations of the edges of the device in the sense of shape sensitivity, therefore, an optimal shape of the cavity is reached after an Lagragian iterative process. Both strategies are evaluated and compared in numerical simulations, being much more effective the second one. In order to evaluate the validity and robustness of the optimization study, we have carried out fully turbulent and three-dimensional numerical simulations at two different conditions defined by their Reynolds Number, \( Re = (U D/\nu) = 2000 \), \( Re = 2 \times 10^4 \), where \( \nu \) is the fluid kinematic viscosity, \( U \) is the incoming flow velocity and \( H \) the height of the immersed model. The proposed optimal shape for a rear cavity reaches an additional drag reduction around a 22\% for lower \( Re \) and a 30\% for higher \( Re \), in comparison with the model with the original drag reduction device.

In conclusion, results from the present study reveal the practical interest of employing drag sensitivity gradient-based procedures in the design of devices related to the control or optimization the flow around heavy vehicles in terms of reduction of fuel consumption.

Acknowledgements: This work has been supported by the Spanish MECD and under Projects FPU2014/02945 and DPI2017-89746-R respectively.
MULTIPLE SCALE ANALYSIS OF THE FRICITION SATURATION OF THE FORCED RESPONSE IN MISTUNED TURBOMACHINERY BLADED DISKS

J. A. Martin, C. Martel

*Departamento de Matemática Aplicada a la Ingeniería Aeroespacial, Universidad Politécnica de Madrid, – 28040 Madrid, Spain.

E-mail: carlos.martel@upm.es, juanangel.martin@upm.es

The estimation of the final vibration amplitude of a bladed disk is of extreme practical importance; it constitutes an essential information for the prediction of the level of high cycle fatigue of the blades, and for the subsequent estimation of its operative life span. The forced response vibration level is saturated by the nonlinear damping introduced by the friction forces at the interfaces between blade and disk (or at the included dampers). The computation of the final amplitude of the saturated limit cycle oscillation requires to solve a quite complicated nonlinear problem. In the case of a tuned bladed disk this problem can be reduced to a single sector with phase lag boundary conditions that requires to consider many time harmonics in order to capture the details of the nonlinear periodic oscillation that sets in. If the small unavoidable differences among blades (mistuning) are also taken into account, then the situation becomes even more complicated because the solution of the mistuned vibration problem requires to consider not only a single sector but the complete bladed disk. The possibility of applying multiple scales techniques in order to drastically simplify this problem will be discussed in this talk. The idea is to take into account the fact that all relevant effects present (forcing, nonlinear friction damping, and mistuning) are, in most practical situations, small effects that develop in a time scale that is much longer than that associated with the natural elastic vibration frequency of the tuned system. The resulting asymptotically simplified models will be used to analyze the characteristics of the final vibration states for both tuned and mistuned bladed disks.
A CONTINUUM-APPROACH MODEL TO NUMERICALLY SIMULATE THE IN-STENT RESTENOSIS PROCESS

Miguel Angel Martinez $^{1,2}$, Javier Escuer $^1$, Sean McGinty $^3$, Estefania Peña $^{1,2}$

$^1$Aragon Institute of Engineering Research. University of Zaragoza, Spain;
$^2$Centro de Investigación en Red en Bioingeniería, Biomateriales y Nanomedicina, Spain;
$^3$Division of Biomedical Engineering, University of Glasgow, UK

Cardiovascular diseases (CVD) are the leading cause of death and disability in the world. Endovascular devices such as stents and balloons have become the most successful devices to treat advanced atherosclerotic lesions. However, one of the main drawbacks associated with these interventions is the development of restenosis. This process comprises three mechanisms: elastic recoil (in the short term), vessel remodeling and neointimal hyperplasia (in the longer term). The last two are related with smooth muscle cell (SMC) migration and proliferation and extracellular matrix (ECM) deposition. Since the precise mechanisms behind restenosis after stenting, so-called in-stent restenosis (ISR), are still not fully understood, it therefore remains a significant clinical challenge to predict which patients will develop ISR. Experimental studies have established a strong correlation between the level of arterial injury caused by the device and ISR [1]. In recent years, mechanobiological models have emerged to relate mechanics to the complex biological response [2]. Two are the main strategies proposed to study this type of problems: discrete agent based models (ABM) [3] or continuum mathematical models [4]. In this paper, we adopt a continuum approach and develop a model that allows us to simulate the restenosis process following the insertion of a stent into a coronary artery. Diffusion-reaction equations are used for modeling the mass balance between biological species in the arterial wall.

A 2D axisymmetric geometry corresponding to an idealized representation of a straight segment of a healthy coronary artery was considered in all simulations. We consider the use of diffusion-reaction equations to model the mass balance between species in the arterial wall. The main biological species considered to play a key role in the restenosis development are growth factors, matrix metalloproteinases, extracellular matrix, smooth muscle cells and endothelial cells. The arterial wall is modeled as a multilayer structure distinguishing two different domains: intima and media layer. The commercial software package COMSOL Multiphysics 5.3 was used to create the computational geometry and to solve numerically the system of coupled partial differential equations by means of the Finite Element Method (FEM). Experimental data from the literature is used to determine the model parameters. Moreover, a sensitivity analysis has been performed to study the influence of the different parameters on the evolution of the model. The model is able to predict the dynamic events occurring within the arterial wall following damage caused by the stent implantation and the consequent response of all the species. We have observed that initially the damage is maximum close to the stent struts while it decreases as we move away from the struts. At time $t = 300$ days, damage is negligible across the entire computational domain.
Microfluidic devices are usually fabricated with soft materials that tend to deform due to the fluid pressure, what naturally gives rise to coupled fluid-structure interaction problems. Recent experimental and theoretical work on the steady flow in shallow deformable microchannels has shown that the flow rate-pressure drop relation is nonlinear due to the deformation of the upper wall [1,2]. Here, we extend these results by studying the start-up flow both in pressure-controlled and flow-rate-controlled configurations. The characteristic scales and the relevant parameters governing the unsteady flow are first identified, followed by the development of an unsteady lubrication theory assuming that the upper wall can be modeled as an elastic plate under pure bending satisfying the Kirchhoff-Love equation, that is governed by three dimensionless parameters. In the limit of negligible solid and liquid inertia, the fluid pressure satisfies a nonlinear diffusion equation with a dimensionless compliance as the only parameter. Under pressure-controlled conditions, the flow admits a self-similar solution. Our unsteady lubrication theory is validated by performing three-dimensional numerical simulations of the full Navier and Navier-Stokes equations for the elastic wall and the conveyed fluid, respectively. The agreement is excellent when the hypotheses behind the model are satisfied. Unexpectedly, we find fair agreement even in cases where the solid and liquid inertia cannot be neglected.

References

A NUMERICAL ANALYSIS OF MULTICOMPONENT DROPLET VAPORIZATION IN MICROGRAVITY


*Department Ingeniería Térmica y Fluidos, Universidad Carlos III de Madrid, Leganés – 28911 Madrid, Spain.

** Department Mechanical and Aerospace Engineering, University of California, San Diego, La Jolla, CA, USA

E-mail: amillan@ing.uc3m.es

The use of liquid fuels is very convenient in combustion-based transport and industry applications because of their availability, high energy density and easy storage in atmospheric conditions [1]. The chemical energy is released by means of combustion in gaseous phase, hence an endothermic phase change needs to be done before fuel usage. Because of its relevance, liquid fuel vaporization and combustion has been the subject of much research, both for sprays [2, 3] and for individual droplets [4, 5, 6]. Because of rising concerns about the quality of the air in the cities, the Diesel-specific engine emissions of pollutants (especially particulates and NOx), must be drastically reduced, together with global emissions of CO2. While Green-house gases emissions can be reduced by increasing the engines efficiency or using biofuels –mostly ethanol, which can be added in significant amounts to existing fuels-, recent experience shows that more fundamental understanding is needed to reduce particulates and NOx emissions.

As part of a wider effort devoted to increase the basic understanding of droplet combustion, in this work we are concerned with the analysis of the vaporization, without gravity, of an individual droplet in a hot and inert nitrogen environment. Our computational results of a simple-component droplets of n-heptane show a very good agreement with both experiments and other numerical simulations available in the literature.

Also, in this work we considered the vaporization of bi-component droplets of n-dodecane or n-heptane and n-hexadecane. In this case, the large difference of boiling temperatures between the two liquids forming the droplet introduced many difficulties in the analysis. At low ambient temperatures below the boiling temperatures of both fuels, we found a two-stage vaporization in which the droplet temperature reaches a plateau before the less volatile fuel starts evaporating after the droplet runs out of the most volatile liquid. On the other hand, at sufficiently high ambient temperature, the experiments found in the literature demonstrated the appearance of puffing, phenomena that cannot be captured in our 1D model. Nevertheless, our simple model turned out to be useful to provide an explanation of the experimental observation in cases when puffing is present.

References

THE BASIC MECHANICAL PRINCIPLES OF INERTIAL CONTINUOUS VARIABLE TRANSMISSIONS WITH RECTIFIER MECHANISMS

F.J. Morales*, F. G. Benitez**

*AICIA. Transportation Group, Camino de los Descubrimientos, 41092-Sevilla, Spain.

**Transportation Engineering, Faculty of Engineering, Universidad de Sevilla, Camino de los Descubrimientos, 41092-Sevilla, Spain.

E-mail: fmorales2@us.es, benitez@us.es

Continuously variable transmission (CVTs) has been a topic of special interest for long time as it allows variable-power devices to have the engine working at its optimum regimen (either minimum or maximum power consumption as required) and keeping unaltered despite the changes in speed and torque required during operation (i.e. vehicles, power machines).

There is a particular group of CVTs with special characteristics: dynamic CVTs. In these, in addition to act on a control element, as in the rest of kinematic type transmission, the gear ratio also depends on the external conditions to which the transmission is subjected, that is, the transmission ratio is also determined by other variables such as the angular velocity of the input shaft and the load torque exerted on the output shaft. Inertial continuous variable transmissions (I-CVT) is a subset of dynamic CVTs needed of mechanisms to rectify the oscillating motion generated during operation into a one-way rotation in order to transfer power to the output shaft.

The choice of the rectifying system and the characteristics of the components heavily depend on the motion to be rectified and from the oscillating mechanism generator.

Most rectifiers use freewheels to transform the oscillating motion into one-way motion. The operation of this type of rectifier involves periods in which the outer track and inner track of freewheels rotate in opposite directions, resulting in that the relative velocity between tracks is twice the speed of the output shaft of the rectifier. This approach has limitations for sustained applications, making the rectifier the weakest link of this type of transmissions.

This communication presents the influence of the rectifier mechanism in inertial CVTs while keeping the other subsystems of the transmission unaltered. Theoretical, numerical and experimental comparative results on the functioning of an inertial I-CVT design operating with two different rectifier mechanisms are discussed. Encountered advantages and disadvantages, regarding transmission efficiency, are identified for two prototyped CVTs.
POSTER 30

THE CONTRACTION REGIMES OF SUBMERGED PINNED MENISCI

D. Moreno-Boza*, M. Rubio-Rubio*, J. Rivero-Rodríguez**, A. Sevilla*

*Grupo de Mecánica de Fluidos, Universidad Carlos III de Madrid, Av. Universidad 30, 28911 Leganés (Madrid).

**TIPs, Université libre de Bruxelles, Avenue Franklin Roosevelt 50, 1000 Bruxelles, Belgium.

E-mail: damoreno@pa.uc3m.es, mrrubio@ing.uc3m.es, asevilla@ing.uc3m.es, jriveror@ulb.ac.be

The controlled generation of microemulsions and microfoams is of great practical interest in the medical and pharmaceutical industries, and has been successfully implemented in microfluidic platforms. The first evidence of break-up and satellite (daughter) formation induced by the contraction of a pinned bubble (mother) in Stokes flow was provided in [1], where the daughter-to-mother volume ratio was found to be remarkably small. The potential applicability of this flow configuration has been demonstrated recently in [2], by periodically generating attolitre liquid droplets inside a viscous liquid matrix. Therefore, and motivated by the lack of a comprehensive characterization of the problem, the contraction dynamics of a meniscus pinned at a needle and immersed in a quiescent liquid ambient is investigated by means of numerical simulations of the relevant two-phase Navier-Stokes equations and experiments at constant suction flow rate under low-Reynolds-number conditions. Through a systematic parametric study we aim at establishing the critical conditions for the formation of satellite bubbles, and quantifying their volume once produced, in the parametric space spanned by the capillary number and the dimensionless initial volume for fixed Bond numbers.

References

A METHOD FOR MEASURING INTERFACIAL TENSION BETWEEN LIQUIDS OF ALMOST THE SAME DENSITY


*Dpto. de Ingeniería Aeroespacial y Mecánica de Fluidos, Universidad de Sevilla, E-41092 Sevilla, Spain.

**Dpto. Ingeniería Mecánica, Energética y de los Materiales, Universidad de Extremadura, E-06006 Badajoz, Spain.

E-mail: mguadama@unex.es

Drop shape methods are of extensive use for surface tension measurement. However, when the Bond number is low, that is for instance, in microgravity conditions or when working with two fluids with very similar densities, the drop shape becomes insensitive to the surface tension value, and therefore, these methods fail to produce accurate measurements. In this work, we propose a method to measure the interfacial tension in those conditions based on the dynamical response of a liquid bridge. We experimentally study the behavior of a liquid bridge when we produce a small amplitude lateral vibration of one of the supporting disks to determine the resonance frequency. The surface tension is calculated by adjusting the value for which the measured resonance frequency matches the numerically calculated natural one. A similar idea was used by Perales and Meseguer [1], though the vibration of the supporting disk was axial, and the accuracy of the method was not evaluated. First, we assess the accuracy of the new method by studying liquid-liquid interfaces with sufficiently large Bond numbers so that the drop shape techniques are still capable of producing accurate results. In these experiments we used a high speed camera and high level image analysis to study the interface response to the vibration and to determine the resonance frequency. The measurements provided by the new method showed good agreement with those produced by drop shape techniques. Then, we proposed a simplified experimental method and setup to perform the measurements. We used an inexpensive digital camera increasing the exposure time to the maximum. The resonance frequency can be easily determined by the operator at a glance. And at this stage, we used the theoretical results obtained by Sanz and Díez [2] to determine the interfacial tension from the resonance frequency. We performed measurements for liquid-liquid interfaces with density differences for which the drop shape methods are known to fail.

References

INITIALIZATION AND RECONSTRUCTION OF IMPLICITLY-DEFINED MULTI-MATERIAL INTERFACES ON NON-CONVEX GRIDS


* Dept. de Ingeniería Mecánica, Materiales y Fabricación, Universidad Politécnica de Cartagena, 30202 Cartagena, Spain
** Dept. de Mecánica, UNED, 28040 Madrid, Spain.
E-mail: jhernandez@ind.uned.es, joaquin.lopez@upct.es

Solving multi-material interface problems with volume-tracking schemes requires the computation of the initial material volume and interfacial surface reconstruction in each cell of the computational mesh, which may be complex and computationally expensive on non-convex geometries. Some authors use a recursive local grid refinement of the cell that has to be initialized, coupled with a low-order (generally linear) representation of the interface and costly decomposition techniques to divide the non-convex regions into multiple convex sub-regions [1]. In this work, we present the application of a set of analytical and geometrical tools [2-5], recently extended to non-convex geometries [5], to the initialization and reconstruction of different interfaces on computational meshes with non-convex cells, without the need for convex decomposition techniques. In particular, (1) a recursive grid refinement method [6] for the computation of the material volume bounded by cell edges and a given implicitly-defined material interface, and (2) different reconstruction methods amenable to any unstructured mesh have been applied on different meshes with non-convex cells and material interfaces with either convex or concave shapes. A detailed assessment of the material-initialization and surface-reconstruction accuracy and the determination of the required computational effort have been carried out. Preliminary results are compared with traditional methods that use convex decomposition techniques in order to show the great computational efficiency achieved with the proposed approach.

The authors gratefully acknowledge the support of the Spanish Ministerio de Economía, Industria y Competitividad and FEDER through project DPI2017-87826-C2.

References

Rheinforce cork composites, previously named elsewhere as CorkSTFluidics, are sandwiches consisting of two microagglomerated cork pads embedding microfluidic patterns that are filled with a shear thickening fluid (see Figure 1 (a)), whose mechanical performance under impact loads is determined by the microfluidic pattern and the rheological properties of the fluid. From the experimental results it is impossible to decouple the fluid contribution from the solid contribution to the force-time response of the composite. In this work we have developed a simple model of the fluid-flow dynamics that it is able to predict the fluid contribution in the energy dissipation of the Rheinforce cork composites under impact loads. To that end, we model the viscous flow inside a microchannel for a known displacement of the upper lid \( h(t, x) \), calculating as a result the needed force to create that movement. The numerical tests for the imposed displacement of the lid followed the law,

\[
h(t_{n+1}) = h(t_n) \cdot (1 - \frac{\Delta t}{T}),
\]

being \( \Delta t \) the time step and \( T \) the characteristic time of the movement. The results are shown in Figure 1 (b), where the analytic prediction developed exhibits an excellent agreement with the numerical tests.

(a) 
(b) 

Figure 1: (a) Detail of the composition of a Rheinforce cork composite, with the microchannels integrated between two microagglomerated cork pads. (b) Temporal evolution of the non-dimensional power needed to move the lid for different values of \( T \) and the comparison with the theory developed.
Cooling towers, among other equipments, could have an important atmospheric impact, becoming a source of pollutants or biological agents. One of these risks is related to the emission of droplets to the atmosphere. These droplets can be a vector for the dispersion of pollutants and infectious agents. The most important, due to its frequency and importance of the outbreaks, is Legionella. In this work, it is presented a new design of mechanical draft cooling tower that limits the emission of droplets to the exterior and prevents the dispersion of Legionella bacteria. This new design is based on drastically lowering the flow velocity at the exit of the tower in order to reduce its capacity to drag drops to the exterior and to keep small droplets generation out of the outlet. To analyze the behavior of this new design, a real prototype has been constructed. It is designed to be able to dissipate a thermal load of 45 kw, is four meters high and covers an area of 12 square meters. The sensitive paper technique is employed to test the capability of this prototype to prevent the emission of droplets to the ambient. Several tests were carried out and no significant emissions were detected. With respect to its thermal behavior, Experimental results show similar values of NTU to commercial mechanical draft cooling towers.
Mechanical force at the tissue level leads to local stress concentrations within the tissue, and, if high enough, starts damaging it at specific spots. In healthy tissues at physiological stress levels, healing continuously repairs such defects to maintain the tissue's structural integrity. Despite increasing experimental and analytical efforts to investigate failure-related irreversible effects of soft biological tissue, the underlying mechanisms are still poorly understood.

The goal of this study was characterize the failure properties of the intact wall and each separated layer (intima, media and adventitia) of the descending thoracic and infrarenal abdominal aorta and to test the hypothesis that the failure properties of layer-separated thoracic arteries differ depending on arterial location in the aorta (Peña et al. 2015). To test this hypothesis, we performed uniaxial tests to study the mechanical behavior of both intact and layer-separated porcine aortic tissue samples taken from descending thoracic and infrarenal abdominal aorta until complete failure.

The damage behavior required a continuum damage theory commonly used to describe the softening of soft tissues under large deformations. The structural model here presented was built within the framework of nonlinear continuum mechanics (Calvo et al. 2007). Tissue damage was simulated considering different damage behaviors for the matrix and the fibers.

We reported values of constitutive parameters using the damage model that can be used by biomedical engineers for investigating better therapies and developing artery-specific devices.

Acknowledgements

Support from the Spanish Ministry of Economy and Competitiveness through the research projects DPI2016-76630-C2-1-R and the Instituto de Salud Carlos III (ISCIII) through the CIBER initiative.

References


Impacts on composites produce interlaminar failure (delamination) which is difficult to detect in common maintenance tasks and affects the structural integrity. Therefore it is critical, for safety, to improve prediction tools in order to perform tolerant damage designs. The numerical modeling of impacts of deformable objects (such as ice) on composite panels is still a challenge. Not only the modeling of the laminate should be appropriate to reproduce its behavior and failures, but also the modeling of the deformable projectile should be capable to induce the corresponding response and damages. In this work simulations of high velocity impacts of ice spheres on carbon/epoxy laminates are accomplished. The Drucker-Prager model has been chosen to describe the mechanical behavior of the ice under high velocity impact conditions. Taking into account the different behavior of ice under tensile and compression and its dependences on the strain rate. The composite material model considers intralaminar and interlaminar failure in order to reproduce the laminate behavior. Results have been validated by means of experimental tests performed in a wide range of impact velocities (50 – 250m/s). The delaminated area was chosen as comparison variable, and reflects that the model predicts adequately the impact process.
MEASUREMENT OF EXTENSIONAL RELAXATION TIMES OF DILUTE VISCOELASTIC POLYMER SOLUTION

A. Ponce-Torres, M. Rubio, E. J. Vega, J. M. Montanero

Departamento de Ingeniería Mecánica, Energética y de los Materiales and Instituto de Computación Científica Avanzada (ICCAEX), Universidad de Extremadura, Badajoz E-06006, Spain.

E-mail: aponce@unex.es

The measurement of extensional rheological properties of viscoelastic solutions is of interest at both fundamental and practical levels. Among those properties, the extensional relaxation time characterizing the capillary thinning has attracted the attention of researchers over the last decades.

The available experimental techniques allow one to measure values down to 100 μs. This lower limit is imposed by the speed of the image acquisition system. In our experiments, a very small viscoelastic pendant drop is inflated until the maximum volume stability limit is reached, and then it spontaneously breaks up. We captured the time evolution of the resulting capillary thinning with an ultra-high-speed camera capable of recording images at up to 5 x 10^6 fps. This allowed us to measure relaxation times of the order of a few microseconds, one order of magnitude smaller than the current experimental limit. We have determined the dependence of the relaxation time upon the concentration of Poly(vinylpyrrolidone) (PVP) far below the overlap concentration. Good agreement has been found between the longest Zimm relaxation time (the relaxation time characterizing a single polymer chain) and the plateau observed for ultra-dilute solutions.
A new numerical method for the simulation of complex, Newtonian and non-Newtonian free-surface flows is presented. The method makes use of a Particle Level Set (PLS) approach along with Adaptive Mesh Refinement (AMR) techniques to retrieve, accurately and efficiently, the fluid interface at each time step as the zero isocontour of a level set function [1]. Marker particles advected by the flow and placed sufficiently close to the interface enhance shape reconstruction, while a second-order accurate reinitialization procedure keeps the level set function from developing numerical instabilities as the simulations go on [2]. The convective terms are dealt with by means of a semi-Lagrangian formulation of the Navier-Stokes equations within a Finite Element framework, leveraging anisotropic mesh refinement techniques developed via error estimation to produce spatially-adapted, ‘optimal’ triangulations. Multiscale simulations of non-Newtonian flows are realized through the kinetic modeling of ensembles of dumbbells scattered over the domain, their internal configurations providing the extra-stress tensor representing the viscoelastic contribution to the Newtonian solvent [3]. The capabilities of the method are illustrated in a series of 2D simulations of pure-advection and complex free-surface flows, showing surface tension and viscoelastic effects.

References

DIFFUSIVE MOLECULAR DYNAMICS SIMULATION OF HYDROGEN DIFFUSION IN MAGNESIUM

J. Ramos*; X. Sun**; M. Ortiz**, M.P. Ariza*

*Department of Continuum Mechanics and Structural Analysis. Universidad de Sevilla
Sevilla, Spain

**Division of Engineering and Applied Science, California Institute of Technology, Pasadena, CA
91125, USA

E-mail: jramos6@us.es, xssun10@caltech.edu, ortiz@caltech.edu, mpariza@us.es

Magnesium (Mg) is a promising material for hydrogen (H) storage due to its low cost, low density and high storage capacity. Understanding the microscopic mechanisms of H mass transport and the induced lattice deformation in Mg is crucial for material designers, since they hold promise to accelerate the loading and unloading processes and increase the energy and power densities.

H diffusion in Mg nanomaterials is a deformation-diffusion coupled problem. Experimental studies [1] have shown that this process can take place over more than 1 hour in Mg nanomaterials with size of 250 nm. The confluence of the atomistic-length scale and such long-time scale poses a significant challenge to the existing atomistic models (e.g., molecular dynamics (MD) and accelerated MD).

The objective of this work is to simulate long-term H diffusion in Mg, while maintaining a fully atomistic description of the materials. To achieve it, we employ a new model, referred to as diffusive molecular dynamics (DMD) [2]. Its basic idea is to couple a discrete kinetic model for the evolution of mass transport process with a non-equilibrium thermodynamics model that governs lattice deformation and supplies the requisite driving forces for kinetics. In this work we will show that the DMD model, when equipped with an angular dependant potential (ADP) function [3], is capable of capturing equilibrium properties and long-term behavior. The dynamics of H diffusion and induced lattice deformation will be described in detail.

References

Within the challenges of this century, the improvement of energy efficiency is undoubtedly one of the most important. But even more, according to the BP Statistical Review of World Energy 2018, global primary energy consumption grew significantly concerning 2017, reporting a record of average growth of 2.2% compared to the previous year. According to the Department of Energy and Climate Change (DECC) declarations that at least 33% of the total CO2 emissions come from the processes of construction and of which 26% come from plants that make use of excavators or backhoes powered by fluids. According to Love, the efficiency of the hydraulic systems is ranged between 9% to 60% showing that their dispersion depends on the application. Mobile hydraulics not only is present in the construction machinery also in agriculture or, especially for urban services where fluids often power the cleaners, small trucks or hydraulic mowers into others.

The paper presents a methodology for conducting energy audits of the hydraulic drives of mobile machinery. In detail, this paper analyzes the energy consumption of urban services machines in especial a refuse truck equipped with side-loading technology. Based on those results, innovative solutions are proposed as independent control, adjustment, recovery of kinetic energy, among others, to achieve optimizations in the consumed energy readings. Moreover, the authors present the indirect kinematic approach based on the image processing of the movement the external mechanical chains. The energy consumption can be estimated knowing the inverse kinematics of the mechanical part complemented with the parameter of the hydraulic system.

References

We study both numerically and experimentally the steady cone-jet mode of electrospaying close to the stability limit of minimum flow rate. The leaky dielectric model is solved for arbitrary values of the relative permittivity and the electrohydrodynamic Reynolds number. The linear stability analysis of the base flows is conducted by calculating their global eigenmodes. The minimum flow rate is determined as that for which the growth factor of the dominant mode becomes positive. We find a good agreement between this theoretical prediction and experimental values. The analysis of the spatial structure of the dominant perturbation may suggest that instability originates in the cone-jet transition region, which shows the local character of the cone-jet mode. The electric relaxation time is considerably smaller than the residence time of a fluid particle in the cone-jet transition region (defined as the region where the surface and bulk intensities are of the same order of magnitude) except for the high-polarity case, where these characteristic times are commensurate with each other. The superficial charge is not relaxed within the cone-jet transition region except for the high-viscosity case, because significant inner electric fields arise in the cone-jet transition region. However, those electric fields are not large enough to invalidate the scaling laws that do not take them into account. Viscosity and polarization forces compete against the driving electric shear stress in the cone-jet transition region for small Reynolds numbers and large relative permittivities, respectively. Capillary forces may also play a significant role in the minimum flow rate stability limit. The experiments show the noticeable stabilizing effect of the feeding capillary for diameters even two orders of magnitude larger than that of the jet. Stable jets with electrification levels higher than the Rayleigh limit are produced. During the jet break-up, two consecutive liquid blobs may coalesce and form a bigger emitted droplet, probably due to the jet acceleration. The size of droplets exceeds Rayleigh’s prediction owing to the stabilizing effect of both the axial electric field and viscosity.
In addition to the experimental difficulties associated with the very low elastic moduli of hydrogels, the main challenge in characterizing hydrogels or similar materials by nanoindentation lies in the time-dependent mechanical behavior they exhibit. Hydrated materials, hydrogels in particular, have been described using a poroelastic model. When a load is applied to a porous material, the increase in pressure in the pores produces a free fluid movement which is time dependent. The poroelastic model explains this behavior by a characteristic length of the test which, in the case of indentation, is related to the radius of the contact between indenter and sample. The smaller the contact size, the faster the equilibrium response is achieved. This is one of the reasons why it is very interesting to do nanoindentation tests on hydrogels or hydrated materials, in a few minutes an equilibrium situation is reached which could require hours or days in macroscopic tests. This indentation size effect in the elastic response of hydrogels also serves to distinguish this behavior from a purely viscoelastic regime which, in principle, does not depend on the size of the indentation.

In this work, a series of nanoindentation tests in polyacrylamide hydrogels were carried out. The combination of a poroelastic-viscoelastic model is used to uncouple these two behaviors, which will be associated with different deformation mechanisms.
The heating and cooling of buildings represents more than 25% of the global energy consumption. For this reason, interest in low-energy buildings and net-zero energy buildings has grown rapidly in recent years. Low enthalpy geothermal energy is considered as one of the most efficient renewable energy sources to reduce the carbon footprint because, when correctly designed and sized, the resulting geothermal HVAC (heating, ventilation, and air conditioning) systems can provide significant monetary savings and a clean and efficient way of cooling and heating buildings.

When using geothermal energy in an HVAC system, the ground can be used in two different ways. First, in extreme climate zones where only heating/cooling is needed, it can be used as a large source/sink into which the heat is exclusively extracted/injected. On the other hand, in moderate climates where heating is needed in winter and cooling is needed in summer, the ground can be used as a large thermal inertia which is recharged with heat in summer and discharged of heat in winter.

Depending on which way the ground is employed, the presence of a groundwater stream can be profitable or not from an engineering point of view. On one hand, it is beneficial when using the ground as a heat source/sink because it adds a new convective heat transport mechanism to the already existent conductive one, improving thereby the recharging capacity of the ground. On the other hand, if the ground is used as thermal inertia, the presence of a groundwater stream prevents recovering in wintertime the energy stored during the summer.

In the present work the thermal response of a vertical geothermal borehole has been obtained taking into account the groundwater effect. Additionally to the extreme slenderness of the borehole and the slowly varying heat injection rate, the Peclet number based on the radius of the borehole turns out to be small compared to unity. These large disparities in time and length scales have been exploited in the present work by means of matched asymptotic expansion techniques, allowing the derivation of analytical expressions for the sought thermal response of the borehole.

The aforementioned heat transfer problem is similar to a classic fluid-mechanical one, the Oseen's problem and the Stokes’ paradox, in the sense that in both problems two different regions emerge: the closer to the borehole one in which only diffusive terms are important, and the further away one in which convective and diffusive phenomena are relevant. The results obtained by the asymptotic analysis have successfully been compared against a detailed numerical simulation of the underlying heat transfer problem.
FRACTURE ANALYSIS OF AN ULTRA-HIGH-PERFORMANCE FIBRE-REINFORCED CONCRETE UNDER THERMO-MECHANICAL LOADING

José D. Ríos, Antonio Martínez-de la Concha, Héctor Cifuentes

Department of Continuum Mechanics and Structural Analysis. Universidad de Sevilla
Sevilla, Spain
E-mail: jdrios@us.es, delaconcha@us.es, bulte@us.es

Ultra-high-performance fibre-reinforced concrete (UHPFRC) is characterised by having a very dense and brittle concrete matrix, usually reinforced with a large amount of steel fibres to increase its tensile strength and ductility. The high addition of steel fibres and the low porosity leads to obtain significant mechanical and fracture properties. In some applications, where the cracking of concrete must be closely controlled, the UHPFRC might reduce or dispense the use of steel prestressed reinforcement. Depending on the steel fibres used, different mechanical and fracture behaviour will be obtained [1]. Though, there are a large amount of papers focused on the mechanical behaviour [2] and fracture properties [3], no many works have established a connection between mechanical or fracture properties with the pore structure of concrete matrix. Additionally, the ultra-high-performance fibre-reinforced concrete must be capable of bearing simultaneously thermal and mechanical loading. In those cases, the microstructure of matrix has a significant influence on the thermal conductivity of concrete [4] and as a consequence, on generation of thermal gradients in the matrix during the heating which might conduct to thermal degradation or even spalling failure.

In this work has been studied the influence of thermal loading, as well as the addition of steel fibres on the microstructure of an ultra-high-performance fibre-reinforced concrete and the consequences of both on the mechanical and fracture properties at room temperature and 300 °C. The influence of microstructure on the mechanical and fracture properties at room temperature and 300 °C was determined through X-ray computed tomography (CT) scan on samples at room temperature and 300°C. Finally, a relationship between the mechanical and fracture properties of UHPFRC with the influence of the presence of fibres, as well as the temperature exposure in the microstructure was established. From results was observed as the reduction of compressive strength, tensile strength and fracture energy as a result of thermal damage was more abrupt, in relative terms, in case of fibre-reinforced concrete than unreinforced one due to its initial lower porosity. The addition of steel fibres decreased more efficiently the porosity and improved material strength, strains and fracture energy were reached. However, the lower porosity of UHPFRC led to a more significant loss of strength by thermal damage.

References

Low temperature geothermal energy is considered as one of the most efficient and promising renewable energy sources, especially when harvested for the HVAC (heating, ventilation, and air conditioning) system of a building. When correctly designed and sized, these geothermal HVAC systems can provide significant monetary savings and a clean and efficient way of cooling and heating buildings. A geothermal HVAC system is typically conformed by a water-to-water heat pump connected to a geothermal heat exchanger composed of multiple vertical boreholes equipped with, normally, U-shaped pipes. Through these pipes a heat carrying liquid flows and exchanges heat with the surrounding ground. A well designed geothermal heat exchanger avoids an excessive drift in the operating temperatures of the heat carrying liquid that otherwise would negatively affect the energy efficiency of the heat pump, and consequently of the whole HVAC system.

Sometimes the buildings require higher heating and cooling needs than usual, for example during the hottest or coldest weeks of the year. These so-called peak loads determine the maximum and minimum operating temperatures of the geothermal HVAC system. During these transitory regimes the ground temperature perturbations reach distances from the borehole that are comparable to the borehole diameter. Thanks to the extreme slenderness of geothermal boreholes, the resulting heat transfer problem can be analyzed using a two-dimensional model, relegating the vertical coordinate of the borehole to the status of a parameter that connects the multiple temperature planes, which are obtained individually.

The formulated heat transfer problem is then solved by means of Fourier expansions for the harmonic case and the Laplace transform for the transitory case, where both of these methods involve a very similar solution procedure. Given the linearity of the equations, the solution of the model considers the presence of an infinite number of heat sources and poles in each of the pipes, which in fact are the eigenfunctions of the problem. These are typically called multipoles in the literature. The numerous effects of all these multipoles are superposed to obtain the overall temperature field in the borehole and the surrounding ground. This approach is closely related to the work done by Claesson and Hellström (2011), who developed the use of multipoles for the steady state case of the problem and were able to obtain solutions with outstanding accuracies. By analyzing the harmonic and transitory regimes, the present work expands their analysis to all relevant time regimes of the problem.

The cylindrical wall boundary layer of a closed cylinder split in two halves at the equator is studied experimentally. When these two parts rotate in exact corotation the internal flow is essentially in solid-body rotation at the angular velocity of both halves. When a slight difference between the rotation frequencies is established a secondary flow is created due to the differential rotation between both sides and restricted to the boundary layer. This behavior of the boundary layer is compared with theoretical and numerical results finding the “sandwich” structure of a Stewartson boundary layer. Time-dependent waves are observed near the cylindrical wall. Their behavior for different values of the control parameters are presented. Finally, a global recirculation mode is also found due to a symmetry-breaking induced between sides that appears because of a slight misalignment of the experimental setup, whose characteristics are compatible with the behavior of a precessing cylinder.
This paper proposes a novel boundary element approach formulated on the Bézier-Bernstein basis to yield a geometry-independent field approximation. The proposed method is geometrically based on both computer aid design (CAD) and isogeometric analysis (IGA), but field variables are independently approximated from the geometry. The proposed formulation shares some characteristics with the standard collocation methods and isogeometric analysis (IGA), exploiting the advantages of each method.

The proposed strategy can be understood as a formulation that lies between the standard collocation method and the isogeometric analysis. The main thrust of this formulation is the use of an independent basis to represent the geometry and the field variables. Thus, the polynomial basis can be used to represent the field variable adequately, preserving the exact representation of the geometry. In this work, we started with the idea that a B-spline can be converted to Bézier curve using the Boehm algorithm. Therefore, taking this identity into account, the proposed method will be derived in the Bézier-Bernstein space.

This approach allows the appropriate approximation functions for the geometry and variable field to be chosen. We use the Bézier-Bernstein form of a polynomial as an approximation basis to represent both geometry and field variables. The solution of the element interpolation problem in the Bézier-Bernstein space defines generalised Lagrange interpolation functions that are used as element shape functions. The resulting Bernstein-Vandermonde matrix related to the Bézier-Bernstein interpolation problem is inverted using the Newton-Bernstein algorithm. The applicability of the proposed method is demonstrated solving wave propagation problems in three-dimensional (3D) and two-and-a-half dimensional (2.5D) domains.
GLASS MICRONOZZLES TO STUDY CELL-DEFORMABILITY AS BIOMARKER

A. Rubio¹, V. Faustino²,³,⁴, R. Agujetas¹, M. G. Cabezas¹, R. Lima²,³ & E. J. Vega¹

¹Depto. de Ingeniería Mecánica, Energética y de los Materiales and Instituto de Computación Científica Avanzada (ICCAEx), Universidad de Extremadura, E-06006 Badajoz, Spain.

²MEtRICs, Mechanical Engineering Department, University of Minho, Campus de Azurém, 4800-058 Guimarães, Portugal.

³Transport Phenomena Research Center (CEFT), FEUP, R. Dr. Roberto Frias, 4200-465 Porto, Portugal.

⁴Microelectromechanical Systems Research Unit (CMEMS-UMinho), University of Minho, Campus de Azurém, 4800-058 Guimarães, Portugal.

E-mail: ejvega@unex.es

Usually, in the literature, a 2D micro-system made of polydimethylsiloxane (PDMS) is used to study cell (RBCs) deformability. The manufacture of these 2D PDMS micro-devices is relatively complex, expensive, and slow, making difficult the mass production of this type of micro-devices, from an industrial point of view. Besides, clogging troubles are common in these micro-devices, limiting its reuse. In this work, we present a new micro-device to assess the deformability of microentities, consisting on a simple axisymmetric borosilicate micronozzle, whose fabrication is simple, fast, and low-cost by using a fire shaping method, allowing large-scale production. A proper quasi-extensional flow is developed in the micronozzle to produce a controlled elongation of microparticles/cells to measure the deformation index (DI). We apply our micro-device to measure and compare the DI of healthy and pathological (chemically treated or artificially impaired) human RBCs, overcoming clogging troubles. The healthy RBCs show a much higher ability to deform that the pathological ones (who turn into more rigid). The comparison between both results allows one to detect with sensitivity enough and diagnose a disease/pathology, by using the DI as a biomarker. Thus, the use of this type of micronozzles have a great potential to detect small changes of cancer and blood cell mechanical properties. The proposed microfluidic device can be easily transformed into a clinical tool for early detection and diagnosis of blood diseases than to its simplicity and low-cost.
The pinching of a fluid-liquid interface is a ubiquitous and spontaneous phenomenon that offers a unique opportunity to observe the behavior of fluids with arbitrarily small length and time scales. This finite-time singularity challenges our natural expectations, normally based on experiments characterized by much larger spatio-temporal dimensions. The pinching of a Newtonian liquid free surface has been profusely analyzed over the last three decades both theoretically and experimentally. Attention has been paid to the thinning of the liquid thread very close to the pinching, where the spatial and time scales are small enough for the system to adopt a universal behavior affected by its intrinsic properties but independent from the boundary and initial conditions.

In this work, we determine experimentally, with an unprecedented spatio-temporal resolution, the evolution of the liquid free surface during the pinching of a pendant droplet. We observe, for the first time, the transition from the inertio-capillary regime to the inertio-viscous-capillary final stage for a low viscosity liquid. Experiments with 5-cSt silicone oil show the formation of a micrometer subsatellite droplet near the pinching region. This phenomenon does not occur for a glycerine/water mixture with the same Ohnesorge number, which indicates the existence of non-Newtonian effects during the breakup of the silicone oil droplet. The comparison between the experimental results and numerical simulations of the full Navier-Stokes equations shows that experiments deviate from the Newtonian theoretical predictions when the subsatellite droplet forms. Viscoelasticity may explain that deviation.
The addition of surfactant confers a certain degree of complexity on Newtonian liquids, which may lead to unexpected behaviors during the pinch-off of their free surfaces. We examine both theoretically and experimentally the breakup of a pendant drop loaded with surfactant. The surfactant is treated as insoluble in our analysis because its characteristic surfactant absorption/desorption time is much bigger than the time characterizing the experiment. We consider the effect of not only solutocapillarity and Marangoni stresses, but also that of surface shear and dilatational viscosities. Experiments are conducted with very high spatial and temporal resolutions, which allows measuring the neck radius on the submicrometer scale. We have conducted both experiments and simulations of several fluids to analyze the relative importance of solutocapillarity, Marangoni stresses and surface shear and dilatational viscosities.
Isothermal bubbly flows are present in many natural processes and industrial operations, where the bubble size may evolve due to coalescence or breakage. In these cases, it is important to predict the bubble size distribution that controls the mass, momentum and energy interfacial transfer. The purpose of the present work is to study the vertical evolution of the bubble size distribution of a bubble swarm confined in a two-dimensional vertical thin-gap cell (1 mm thickness, 800 mm high, 400 mm width) filled with distilled water. In particular, we analyze the contribution of the gravity driven mechanisms to coalescence.

The proposed confined flow configuration enables the detailed analysis of the coalescence process, since confinement not only favors coalescence, but also its observation, because it does not allow the bubbles to overtake each other out of the plane of the cell. To analyze the coalescence, evolving statistics of the population of bubbles as they rise in distilled water are explored from uncorrelated shadowgraph images of the bubble swarm at several heights. Every image is digitally analyzed using an in-home developed image processing algorithm, which detects each bubble, obtaining its size and position. Moreover, every bubble collision, previous to coalescence, can be identified, obtaining the relative angular position of the involved bubbles.

In order to assess the role of the initial number of bubbles per unit volume, systematic experiments were performed for different values of the air volume fraction. Considering that the swarm is generated with bubbles of the same diameter, the downstream evolution of the bubble size distribution is analyzed by performing a characterization of the area occupied by bubbles of size $d_B$ compared to that of the entire distribution through the bubble projected area probability density function, $A_{pdf}$. Major changes of the $A_{pdf}$ are obtained along the vertical direction, which are mainly caused by bubble coalescence. Moreover, it has been observed that stronger wakes and vortex shedding develop as larger bubbles are generated. Therefore, a higher agitation in the liquid phase is induced, leading to new interactions driven by vortex induced- and wake entrainment-mechanisms. Finally, these mechanisms have been directly studied through a detailed analysis of the relative angular position and the size of the bubbles involved in the collisions. It reveals that they take place mainly between bubbles of different sizes, with smaller bubbles entrapped by the wake of larger ones.

Supported by the Spanish MINECO grants DPI2014-59292-C3-3-P and DPI2017-88201-C3-2-R and by INPT Toulouse SMI 2018.
POSTER 52

DRAG RECONFIGURATION OF ELASTIC PLATES TOWED NEAR THE FREE SURFACE

S. Satheesh, F.J. Huera-Huarte

Department of Mechanical Engineering, Universitat Rovira i Virgili, Av. Països Catalans 26, 43007 Tarragona, Spain.

E-mail: francisco.huera@urv.cat

Plates of different flexural rigidities were towed normal to the flow at a fixed speed in a quiescent fluid to study the effects of the proximity to the free surface on the fluid loading seen by the structure. Direct force measurements were performed in a towing tank set-up that allowed the quantification of the wake dynamics using planar Particle Image Velocimetry (PIV).

Flexibility led to drag reduction due to the reconfiguration of the plates, at all submergence depths, with certain plates exhibiting depth-independent behaviour. The problem was studied parametrically for a constant Reynolds number and plate aspect ratio but covering a wide range of values of other governing dimensionless parameters such as submergence depths, drag coefficients, Cauchy and Reconfiguration numbers. It was observed how the reconfiguration process heavily altered the gap flow formed between the free surface and the plate yielding large shear layer modifications.

The study of Vogel exponents showed that a sub-linear or even a depth independent relationship between drag and velocity takes place for certain Cauchy numbers.

Fig. 1 Variation of the Reconfiguration number with Cauchy number for different submergence depths
GENERALIZED CONTINUUM MODEL FOR THE ANALYSIS OF NONLINEAR VIBRATIONS IN TAUT STRINGS

Ó. Serrano*, R. Zaera*, J. Fernández-Sáez*, M. Ruzzene**


**Vibration and Wave Propagation Laboratory, Georgia Institute of Technology, 275 Ferst Dr, Atlanta, GA 30332, USA.

E-mail: oscar.serrano@uc3m.es, ramon.zaera@uc3m.es, jose.fernandez@uc3m.es, ruzzene@gatech.edu

Classical continuum models are unable to capture scale effects. In dynamics of microstructured solids this effect appears when they undergo vibrations whose wavelength is of the order of the characteristic length of the microstructure. In contrast, discrete models can capture the dynamic behavior but at expenses of high computational cost. Alternatively, generalized continuum models can be considered to meet the commitment of capturing scale effects and computational efficiency.

In this work we formulate an inertia gradient nonlinear generalized continuum model [1] of a taut string from a discrete chain of particles linked to first neighbors by linear springs undergoing large-amplitude (nonlinear) longitudinal and transversal vibrations. Unlike the corresponding classical continuum model [2], an enrichment of the kinetic energy density with the gradient of the velocities and the characteristic length of the microstructure permits the model to capture short wavelength vibrations, regardless the displacement amplitude. Fig. 1 compares the dispersion relations (a) and the frequency versus the initial amplitude of the vibration (b) of the discrete, classical continuum and generalized models. It can be observed that the generalized model provides a better approximation of the dynamic behavior of the discrete model as compared to the classical one.

Figure 1: (a) Dispersion relation, (b) Frequency versus initial amplitude of the vibration.

References

**Poster 54**

**Diffusion-Driven Growth of a Bubble Cloud in a Supersaturated Liquid in Microgravity**

P. Vega-Martínez*, J. Rodríguez-Rodríguez, D. van der Meer**, M. Sperl***

* Fluids Mechanics Group. Universidad Carlos III de Madrid, Leganes, Madrid, Spain,

** Physics of Fluids Group, MESA+ Research Institute, and J.M: Burguers Center for Fluids Dynamics, University of Twente, P.O.Box 217, 7500, AE Enschede, The Netherlands

*** Institut für Materialphysik im Weltraum, DeutschesZentrum für Luft- und Raumfahrt, 51170 Cologne, Germany.

E-mail: pvega@ing.uc3m.es

Diffusion-driven growth of a foam ball appears in many manufacturing processes as well as in a variety of geological phenomena, i.e. the formation of rocks in meteorites and other small celestial bodies. In Earth, bubble growth is highly affected by gravity, becoming dominant after hundreds of milliseconds. The aim of this research is to explore experimentally the diffusion-driven growth of a bubble cloud in a gas-supersaturated liquid in microgravity conditions. Here, we describe the experiments carried out in the drop tower of the Center of Applied Space Technology and Microgravity (ZARM). In these experiments, we are able to observed the evolution of the bubble cloud and the collective effects of bubbles for 3 seconds in microgravity.
POSTER 55

NEAR-LIMIT FINGER-LIKE PREMIXED HYDROGEN-AIR FLAMES PROPAGATION IN NARROW CHANNELS

F. Veiga-López*, D. Martínez-Ruiz**, M. Kuznetsov***, M. Sánchez-Sanz*

*Depto. de Ing. Térmica y de Fluidos, Universidad Carlos III de Madrid, 28911, Leganés, Madrid, España

**Depto. de Mec. De Fluidos y Prop. Aeroespacial, Univ. Politécnica de Madrid, 28040, Madrid, España

***Institut für Kern- und Energietechnik, Karlsruhe Institut für Technologie, 76344, Eggenstein-Leopoldshafen, Deutschland

An experimental study of lean hydrogen-air premixed flames in the range of equivalence ratio from 0.2 to 0.51 propagating in a confined combustion chamber is presented in this poster. The combustion chamber has a rectangular prismatic shape of 900 x 200 x (4-1) mm (length x width x thickness). The thickness of the channel can be modified to study its effect. Furthermore, the ignition point can be shifted from top to bottom to assess the impact of gravity. All the flames travel from the open end of the chamber towards its closed opposite side.

Under such conditions, the interplay between heat losses, viscous friction and acoustics determines the propagation of the flames. When testing mixtures close to the flammability limit (influenced by the geometry and orientation of the vessel), the flame advances forming characteristic finger-like shapes presenting big quenched areas in between them.

The tested flames can form isolated cells that travel in straight direction at a constant velocity until extinction and show a high-curvature reaction front. The flame cells are formed by one or two heads provided the ignition position.

In slightly richer mixtures, a different behavior is found. The flames bifurcate into several small branches that travel in transverse direction to the main flame cell, propagating as fractals with a characteristic fractal dimension of around 1.6-1.8. In this case, both one and two-headed flames can coexist, forming characteristic chaotic patterns that remind of a snowflake or a tree.