



# ABSTRACT BOOK

Edited by Professor Dr.-Ing. Andreas Öchsner, D.Sc. (ACEX-Chair)  
**6<sup>th</sup> International Conference on Advanced Computational  
Engineering and Experimenting – ACEX2012**

Istanbul, Turkey • 01-04 July, 2012

[www.acex-conference.com](http://www.acex-conference.com)



**ACE-X2012 • ISTANBUL**  
1-4 July Turkey

.....  
IRONIX CONFERENCE MANAGEMENT

*We organise conferences*

Rua Prior Urbino de Pinho, 2, 2DT

3830-126 Ilhavo, PORTUGAL

[www.ironix-conferences.com](http://www.ironix-conferences.com)



## PREFACE

It is our great pleasure to welcome you to the **6<sup>th</sup> International Conference on Advanced Computational Engineering and Experimenting**, ACE-X 2012, Istanbul, Turkey, from 1-4 JULY, 2012.

ACEX conferences aimed at attracting a balanced portion of delegates from academia, industry and research institutions and laboratories involved with research and development work. In doing so, the conference provides a binding platform for academics and industrialists to network together, exchange ideas, provide new information and give new insights into overcoming the current challenges which are faced by the academics and the industrialists related to the computational engineering and related experimenting.

I would like to thank the Organising Committee members and members of the Local Committee for their help in contributing to the successful organisation of this meeting.

I would like to thank the colleagues, organisers of the SPECIAL SESSIONS, Thank you!

Thanks to all delegates for the decision in attending ACEX2012 and I hope you will find the meeting very useful for your work, business and a useful forum for obtaining new knowledge.

Have fun learning and meeting new people!

**Professor Andreas Öchsner**

ACEX-CONFERENCE – Chairman

Looking forward to welcome you at ACEX2013 in Madrid (Spain)!



## **CFD Simulations of Flow in Capillary Flow Liquid Acquisition Device Channel**

**John B. McQuillen 1, a, David F. Chao1,b, Nancy R. Hall1,c, Michael P. Doherty1,d, Nengli Zhang2,e,†**

1NASA Glenn Research Center, Cleveland, OH 44135, U.S.A.

2Ohio Aerospace Institute at NASA Glenn Research Center, Cleveland, OH 44135, U.S.A.

ajohn.b.mcquillen@nasa.gov, b david.f.chao@nasa.gov, cnancy.r.hall@nasa.gov,

dmichael.p.doherty@nasa.gov, enzhang@grc.nasa.gov

A capillary flow liquid acquisition device (LAD) for cryogenic propellants has been developed and tested in NASA Glenn Research Center to meet the requirements of transferring cryogenic liquid propellants from storage tanks to an engine in low gravity or the reduced acceleration environments. Ideally, it should not limit the flow in normal gravity during liftoff and first-stage flight. The prototypical mesh screen channel LAD was fabricated with a mesh screen, covering a rectangular flow channel with a cylindrical outlet tube, and was tested with liquid oxygen (LOX). In order to better understand the performance in various gravity environments and orientations with different submersion depths of the screen channel LAD, a series of computational fluid dynamics (CFD) simulations of LOX flow through the LAD screen channel was undertaken. An important concern for the performance of the screen channel is the orientation and submersion depth effects of the LAD channel in normal gravity environment on the pressure and flow fields in the channel, and the mass flow rate passing through the channel. A series of simulations for horizontally and vertically submersions of the LAD channel assembly at normal gravity environment was conducted. The gravity effects on the flow in the screen channel have been revealed.

ACEX137

Mr. Saif Akram

Aligarh Muslim University,  
India

## **Natural Convection in Enclosures under the Conditions of Constant Heat Flux and Constant Wall Temperature**

**Saif Akram**

Student of M.Tech.- Department of Mechanical Engineering, Zakir Hussain  
College of Engineering and Technology, Aligarh Muslim University, Aligarh, India

Numerical analysis has been carried out for two-dimensional unsteady natural convective heat transfer in a rectangular enclosure filled with air under the conditions of constant heat flux and temperature at one wall (active wall) while the cold wall is maintained at a fixed temperature. Solutions are obtained for Rayleigh number values from 104 to 106 and inclination angle 90° (vertical enclosure). The non-dimensional temperature and stream functions within the enclosure are obtained for different conditions and plotted for discussion. A comparative study for different parameters under the conditions of constant heat flux and constant wall temperature has been presented in detail. The references are as follows:-

- [1] Oosthuizen, P. H. And Naylor, D., Introduction to Convective Heat Transfer Analysis, William C. Brown Pub., (1998)
- [2] Ede, A. J. "Advanced In Natural Convection" Adv. Heat Transfer, vol 4, (1967).
- [3] Shihe Xins, Patrick Lee Quere "Natural Convective Flow in Air Filled, Differentially Heated Cavities with Adiabatic Horizontal Walls", Numerical Heat Transfer, Part A; 50; 436-466, (2006).

ACEX193

Prof. A. Alberto Torres Garcia Portugal

Universidade de Coimbra,  
Portugal

## **Simulation of two-dimensional combustion propagation on Fe<sub>2</sub>O<sub>3</sub>/Al thermite non-**

## homogeneous disk-shaped systems

P. Brito<sup>1,2</sup>, L. Durães<sup>1</sup>, A. Portugal<sup>1</sup>

1 CIEPQPF, Dept. of Chem. Engineering., Faculty of Sciences & Technol., University of Coimbra, Pólo II, Rua Sílvio Lima, 3030-790 Coimbra - Portugal

2 Dept. of Chem. & Biolog. Technol., School of Technol. & Management, Polytech. Inst. of Bragança, Campus de Santa Apolónia, PO Box 1134, 5301-857 Bragança - Portugal

In previous works[1-2], a two-dimensional model was constructed and tested in a successful attempt to simulate the non-steady radial combustion propagation on thin disk shaped samples of Fe<sub>2</sub>O<sub>3</sub>/Aluminum thermite mixtures. The main characteristics of the model can be summarized as: pseudo-homogeneous medium considering the contribution of air to simulate a porous medium; zero order, non-temperature dependent beyond a given threshold kinetics; conductive/radiative heat transfer; phase transitions; temperature and composition variation of all system properties during propagation; non-homogeneous and randomly generated mixing of reactants on radial and angular directions. Hence, an adaptive numerical algorithm that combines a Adaptive Method of Lines (AMOL) strategy based on finite differences non-uniform spatial discretizations, with collocation schemes based on sets of increasing level dyadic grids, is applied for the simulation of the system.

This particular integration method proves to handle adequately with the steep travelling thermal wave in 2D spatial domain supports, either for trivial non-homogeneous mixing conditions, as for the replication of more sophisticated non-uniformities in the reactant medium that are usually observed in experimental conditions, such as the local lack of reactants due to the presence of an obstacle, and density or thickness variation.

[1] P. Brito, L. Durães, A. Portugal, ACE-X 2011, Vilamoura, Portugal, p. 189 (2011).

[2] P. Brito, L. Durães, A. Portugal, Comp. Thermal Sci., (2012) – submitted for publication.

ACEX206

Mr. Mengdi Song

Université de Technologie de Compiègne,  
France

## Numerical Simulation using Fluid-Structure Coupling Method Applied to an Undulating-Membrane Pump

M.D. Song<sup>1</sup>, E. Lefrançois<sup>1</sup>, M. Rachik<sup>1</sup>

1 Laboratoire Roberval UMR 6235, Université de Technologie de Compiègne, Compiègne, 60205, France.

A sequential coupling method for the simulation of fluid-structure interaction problem (FSI) is proposed in this paper. The main idea is to improve the efficiency of calculations and to offer a better understanding about the functions of an innovative pump which was devised and made by our partner, the company AMS R&D. This new design of pump will produce the propulsion of fluid with an axisymmetric undulating rubber membrane (Active Membrane Systems, AMS® [1]).

The adopted sequentially coupled approach uses three distinct computational codes and the coupling process is as follows:

1. The structure code is designed to calculate the dynamic response of the displacement for the membrane and to simulate the transmission of kinetic energy from the structure to the fluid. Meanwhile, a forced displacement is imposed on the outline of the membrane as boundary condition.

2. The fluid code based on moving mesh technique is designed to estimate the velocity field and the pressure field involved by the displacement of the membrane. Due to the assumptions of an incompressible and inviscid fluid, the velocity field can be determined by Laplace's equation of the velocity potential, while the pressure field can be figured out by Bernoulli equation for unsteady potential flow [2]. This simple approach adapts perfectly to our application (the effect of concentric piston), it is therefore unnecessary to solve Navier-Stokes equations.

3. A fluid mesh deformation at each time steps is processed by the pseudo-material analogy [3] in order to keep the kinematic compatibility between the fluid mesh boundaries and the associated moving structure. This sequential coupling scheme requires the data communications between the different codes to ensure the regular updating of coupled variables. To show the reasonableness and the high performance of proposed method, we first validate our method compared to an