INCAS – National Institute for Aerospace Research “Elie Carafoli”
(under the Aegis of the Romanian Academy)

Proceedings
of
the 36th “Caius Iacob” Conference
on
Fluid Mechanics and its Technical Applications

29 – 30 October, 2015
Bucharest, Romania

Extended Abstracts

Organizers

INCAS – National Institute for Aerospace Research
“Elie Carafoli”
(under the Aegis of the Romanian Academy)

ISMMA – Institute of Mathematical Statistics and
Applied Mathematics of the Romanian Academy
“Gheorghe Mihoc - Caius Iacob”

University of Bucharest

“Politehnica” University of Bucharest

BUCHAREST
2016

Proceedings
of
the 36th “Caius Iacob” Conference
on
Fluid Mechanics and its Technical Applications

29 – 30 October, 2015, Bucharest, Romania

Sessions:

Aerodynamics Design; Numerical Analysis;
Equations of Mathematical Physics; Mathematical Modelling

Editorial Board – Scientific Committee:

Prof. Dr. Corneliu BALAN – POLITEHNICA” University of Bucharest, Power Engineering Faculty, Bucharest, Romania
Dr. Valentin BUTOESCU – INCAS – National Institute for Aerospace Research “Elie Carafoli”, Bucharest, Romania
Prof. Dr. Adrian CARABINEANU – University of Bucharest, Bucharest, Romania
Prof. Dr. Sterian DANAILA – POLITEHNICA” University of Bucharest, Faculty of Aerospace Engineering, Bucharest, Romania
Dr. Stelian ION – ISMMA – Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy “Gheorghe Mihoc - Caius Iacob”, Bucharest, Romania
Dr. Victor MANOLIU – AEROSPACE CONSULTING, B-dul Iuliu Maniu 220, Bucharest, Romania
Prof. Dr. Alexandru M. MOREGA – “POLITEHNICA” University of Bucharest, Faculty of Electrical Engineering, Splaiul Independentei 313, 060042 Bucharest, Romania;
Dr. Florin MUNTEANU – AEROSPACE CONSULTING, B-dul Iuliu Maniu 220, Bucharest, Romania
Dr. Catalin NAE – INCAS – National Institute for Aerospace Research “Elie Carafoli”, Bucharest, Romania
Dr. Sorin RADNEF – INCAS – National Institute for Aerospace Research “Elie Carafoli”, Bucharest, Romania
Dr. George SAVU – COMOTI – National Research and Development Institute for Gas Turbines Bucharest, Romania

Editing:

Prog. Elena NEBANCEA − INCAS – National Institute for Aerospace Research “Elie Carafoli”, Bucharest, Romania

Graphic cover:

Eng. Emil COSTEA − INCAS – National Institute for Aerospace Research “Elie Carafoli”, Bucharest, Romania

Publisher: **INCAS – National Institute for Aerospace Research “Elie Carafoli”**

Copyright©INCAS, 2016. All rights reserved.

***Registration code*:** **ISSN 2067-4414**
ISSN-L 2067-4414
ISSN National Center
Romanian National Library

INCAS – National Institute for Aerospace Research “Elie Carafoli”
(under the Aegis of the Romanian Academy)

Proceedings
of
the 36th “Caius Iacob” Conference
on
Fluid Mechanics and its Technical Applications

29 – 30 October, 2015
Bucharest, Romania

Extended Abstracts

Organizers

INCAS – National Institute for Aerospace Research
“Elie Carafoli”
(under the Aegis of the Romanian Academy)

ISMMA – Institute of Mathematical Statistics and
Applied Mathematics of the Romanian Academy
“Gheorghe Mihoc - Caius Iacob”

University of Bucharest

“Politehnica” University of Bucharest

BUCHAREST
2016

Publisher: **INCAS – National Institute for Aerospace Research "Elie Carafoli"**
B-dul Iuliu Maniu 220, 061126 Bucharest, Romania
E-mail: incas@incas.ro; http://www.incas.ro
Contact person: Elena NEBANCEA

Proceedings of the 36th "Caius Iacob" Conference on Fluid Mechanics and its Technical Applications **ISSN 2067-4414**
29 – 30 October, 2015, Bucharest, Romania

Institutul Național de Cercetări Aerospațiale "Elie Carafoli"

INSTITUTUL DE STATISTICA MATEMATICA SI MATEMATICA APLICATA "GHEORGHE MIHOC-CAIUS IACOB"

UNIVERSITATEA DIN BUCUREȘTI

Universitatea Politehnica București

The 36th "Caius Iacob" Conference
of
Fluid Mechanics and its Technical Applications

Programme

General Schedule

Thursday 29 October 2015

08.30 - 10.00	**Registration**
10.00 - 10.30	**Opening Ceremony** *In Memoriam Mircea Cazacu*
10.30 - 11.10	**Plenary Lectures**
11.10 - 11.30	*Coffee Break*
11.30 - 13.10	**Plenary Lectures**
13.10 - 14.00	*Lunch*
14.00 - 15.20	**Communications**
15.20 - 15.40	*Coffee Break*
15.40 - 17.00	**Communications**
17.00 - 17.20	*Coffee Break*
17.20 - 18.40	**Communications**
19.00 -	*Banquet*

Friday 30 October 2015

09.00 - 09.45	**The "Caius Iacob" Prize Award Ceremony,** **The "Nicolae Tipei" Prize Award Ceremony**
09.45 - 10.00	*Coffee Break*
10.00 - 11.20	**Plenary Lectures**
11.20 - 11.40	*Coffee Break*
11.40 - 13.00	**Plenary Lectures**
13.00 - 14.00	*Lunch*
14.00 - 16.00	**Communications**
16.00 -	**Closing Ceremony**

Plenary Lectures

Thursday		**Plenary Lectures**
	“Elie Carafoli” Amphitheatre	
	Chairman: Cătălin NAE, Cornel OPRIŞIU	
10.30 - 11.10	**Mihai ARGHIR**	*Synthesis of Squeeze Film Damper Numerical Analyses and Comparisons with Experimental Results*
11.30 - 12.10	**Gabriela MARINOSCHI**	*An inverse problem for recovering the diffusion coefficient in a parabolic diffusion equation*
12.10 - 12.50	**Adriana NĂSTASE**	*Comparison of Critical Behaviors of Elliptic and Hyperbolic Quadratical Algebraic Equations with Variable Coefficients*
12.50 - 13.10	**Séverine DUBROECQ**	*Latest developments in laser technics for experimental Fluid Mechanics*

Friday		**Plenary Lectures**
	“Elie Carafoli” Amphitheatre	
	Chairman: Sorin RADNEF, Adriana NĂSTASE	
10.00 - 10.40	**Fahad RIAZ**	*Development of Framework for Acquisition of Avionics Integration Capability*
10.40 - 11.20	**Dan MATEESCU**	*Unsteady effects due to unsteady flow separations at low Reynolds numbers for stationary airfoils*
11.40 - 12.20	**REOROM Group and Corneliu BĂLAN**	*Flow visualizations at low and very low Reynolds numbers*
12.20 - 13.00	**Alexandru MOREGA**	*Heat Transfer Fluid-Flow Interactions in Medical Procedures and Techniques*

Thursday		**Workshop: Numerical Analysis**
“Nicolae Cămărăşescu” Room		
Chairman: Stelian ION, Marius STOIA-DJESKA, Florin FRUNZULICĂ		
14.00 - 14.25	**Tudorel Petronel AFILIPOAE, Marius STOIA-DJESKA**	*Development of a Three-Dimensional Unstructured Euler Solver for High-Speed Flows*
14.25 - 14.50	**Dumitru PEPELEA, Florin FRUNZULICĂ, Marius Gabriel COJOCARU**	*Supersonic flow over bidimensional model*
14.50 - 15.15	**Mihai Leonida NICULESCU, Marius Gabriel COJOCARU, Dumitru PEPELEA, Mihăiţă Gilbert STOICAN, Maria Cristina FADGYAS**	*Computations of the axisymmetric flow with shock waves at hypersonic velocities taken into account the chemical reactions that appear in the air at high temperatures*
15.15 - 15.20	**Discussion**	
15.40 - 16.05	**Ionuţ-Cosmin ONCESCU, Alexandru-Mihai CISMILIANU and Florin FRUNZULICĂ**	*An impact study of a capsule with a rigid wall using the SPH approach*
16.05 - 16.30	**Balazs ALBERT**	*Two and Three Dimensional Blood Flow Simulation in Different Types of Blood Vessels*
16.30 - 16.55	**Marius Gabriel COJOCARU, Mihai Leonida NICULESCU, Dumitru PEPELEA**	*Leading Edge Device Aerodynamic Optimization*
16.55 - 17.00	**Discussion**	
17.20 - 17.45	**Bogdan-Alexandru BELEGA**	*Modelling and Simulation Aircraft Ditching Using SPH*
17.45 - 18.10	**J.B. DUMITRU, A.M. MOREGA, M. MOREGA, and L. PÎSLARU-DĂNESCU**	*Flow Patterns in the Magnetic Fluid Core of a Miniature Planar Spiral Transformer*
18.10 - 18.35	**Mihai Victor PRICOP, Marius Gabriel COJOCARU, Maria Cristina FADGYAS, Mihai Leonida NICULESCU, Mihăiţă Gilbert STOICAN, Dumitru PEPELEA**	*Parallel trajectory propagation tool for preliminary mission analysis*
18.35 - 18.40	**Discussion**	

Thursday	Equations of Mathematical Physics	
“Nicolae Tipei” Amphitheatre		
	Chairman: Ştefan BALINT, Gelu PAŞA	
14.00 - 14.20	**Corina CIPU, Victor ŢIGOIU**	*The flow of an Oldroyd-B fluid between two parallel plates with a porous top plate*
14.20 - 14.40	**Raisa PAŞCAN,** **Sanda CLEJA-ŢIGOIU**	*Continuous defects: dislocations and disclinations in finite elasto-plasticity*
14.40 - 15.00	**Ruxandra STAVRE**	*Optimization of a fluid-structure interaction problem*
15.00 - 15.20	**Tudor Corneliu IONESCU,** **Orest Vasile IFTIME,** **Alexandru DUMITRACHE**	*Model reduction by moment matching for boundary control PDEs*
	Chairman: Sanda CLEJA- ŢIGOIU, Ruxandra STAVRE	
15.40 - 16.00	**Isabelle GRUAIS,** **Dan POLIŞEVSCHI,** **Florentina-Alina STĂNESCU**	*The effective permeability of homogenized fractured porous media*
16.00 - 16.20	**Cecil P. GRUNFELD**	*On a class of abstract kinetic equations in ordered spaces*
16.20 - 16.40	**Gheorghe JUNCU,** **Constantin POPA**	*Brinkman – Forchheimer – Darcy Flow past an Impermeable Cylinder Embedded in a Porous Medium*
16.40 - 17.00	**Oana LUPAŞCU**	*Modeling the onset and the fragmentation phase of a snow avalanche*
	Chairman: Dan POLIŞEVSCHI, Victor ŢIGOIU	
17.20 - 17.40	**Agneta M. BALINT,** **Ştefan BALINT**	*Global and local stability and instability of the constant spatially developing gas flow*
17.40 - 18.00	**Adrian CARABINEANU**	*The Numerical Study of the MHD Faraday Generators*
18.00 - 18.20	**Gelu PAŞA**	*Hele-Shaw Displacements without Surface Tensions*
18.20 - 18.40	**Stelian ION,** **Dorin MARINESCU,** **Ştefan-Gicu CRUCEANU**	*Riemann Problem for Shalow Water Equation with Porosity*

Thursday		Mathematical Modeling
	“Elie Carafoli” Amphitheatre	
	Chairman: Gabriela MARINOSCHI, Alexandru MOREGA	
14.00 - 14.20	**Horia DUMITRESCU, Vladimir CARDOŞ, Alexandru DUMITRACHE, Ion MĂLĂEL**	*Turbulent Boundary Layer at Large Re*
14.20 - 14.40	**Iulia - Rodica DAMIAN, Corneliu BĂLAN, Adriana ENACHE**	*Aspect Ratio Influence on Hydrodynamic Focusing Phenomenon in Cross-Junction Microchannels*
14.40 - 15.00	**Claudiu PATRAŞCU, Ioana - Laura OMOCEA, Corneliu BĂLAN**	*Coalescence Phenomenon of Immersed Jets*
15.00 - 15.20	**Alexandru-Mihai CISMILIANU, Alexandru BOROS, Ionuţ-Cosmin ONCESCU, Florin FRUNZULICĂ**	*New Urban Vertical Axis Wind Turbine Design*
	Chairman: Horia DUMITRESCU, Valentin Adrian Jean BUTOESCU	
15.40 - 16.00	**Margit PAP**	*Mathematical modeling in cornea topography, numerical aspects, applications*
16.00 - 16.20	**Norbert ANGI, Angel HUMINIC**	*Preliminary design of a LSA aircraft using wind tunnel tests*
16.20 - 16.40	**Casandra Venera BALAN (PIETREANU)**	*Boolean algebra application in analysis of flight accidents*
16.40 - 17.00	**Corneliu BERBENTE**	*A Hydro-Dynamical Model for Gravity*
	Chairman: Corneliu BERBENTE, Vladimit CARDOŞ	
17.20 - 17.40	**Valentin BUTOESCU, Ştefan NEBANCEA, Mihaela PETRE, Viorel ANGHEL, Ioan STOICA**	*On the Wind and Snow Loads Distribution and Their Effects on a K-Span Steel Arch Structure*
17.40 - 18.00	**Irina Carmen ANDREI, Alexandra STĂNESCU**	*Issues on modeling and simulation of a mixed flows turbofan*
18.00 - 18.20	**Florin MINGIREANU**	*Study of impulsive trajectory correction for GRAD rocket through six degrees of freedom modelling*
18.20 - 18.40	**Florin MINGIREANU**	*Report of high altitude UAV missions during high altitude balloon flights campaign*

Friday		**Aerodynamic Design**
	“Nicolae Tipei” Amphitheatre	
	Chairman: Stelian ION, Ştefan Bogos	
14.00 - 14.20	**Sorin ARSENE,** **Ioan SEBEŞAN**	*The influence of aerodynamic forces on the vehicle bodywork of railway traction*
14.20 - 14.40	**Mihai Victor PRICOP,** **Mihăiţă Gilbert STOICAN,** **Mircea BOSCOIANU**	*A tool for Newton flow model and heat flux computation*
14.40 - 15.00	**Andrei PAVEL,** **Marius Gabriel COJOCARU,** **Ştefan BOGOS,** **Cristina Maria FADGYAS**	*Crosswind Impact on Medium-Haul Aircraft During Ground Roll*
15.00 - 15.20	**Stelian GRĂDINARU**	*Steady flow of an incompressible perfectly conducting fluid past a thin airfoil of non-zero thickness*

Friday		Mathematical Modeling
	"Elie Carafoli" Amphitheatre	
	Chairman: Corneliu BĂLAN, Daniela BARAN	
14.00 - 14.20	**Alexandru DUMITRACHE and Tudor Corneliu IONESCU**	*Determination of the potential flow spectrum in a wind tunnel nozzle by means of singularities method*
14.20 - 14.40	**Dumitru POPESCU, A. G. POPESCU**	*A Possible Application Of Pulsatory Liposome in The Depresion Treatment*
14.40 - 15.00	**Ştefan SIMIONESCU, Corneliu BĂLAN**	*Study of Natural and Forced Heat Transfer Coefficient on a Vertical Heated Plate*
15.00 - 15.20	**Nicolae APOSTOLESCU, Daniela BARAN**	*Sammon mapping for preliminary analysis in Hyperspectral Imagery*
15.20 - 15.40	**Georgiana PĂDURARU, Andrei TUDOR, Alina PETRESCU, Ioan PLOTOG**	*Mechanical performances of lead-free solder joint connections with applications in aerospace domain*

CONTENTS

Note:

- *Please, note that the responsibility for the accuracy of expression in English belongs to the authors.*

PLENARY LECTURES

Synthesis of Squeeze Film Damper Numerical Analyses and Comparisons with Experimental Results

Mihai ARGHIR*

*Corresponding author
Institut PPRIME, UPR CNRS 3346, Université de Poitiers, ISAE ENSMA,
11 Bd. Pierre et Marie Curie, 86962 Futuroscope Chasseneuil, France
mihai.arghir@univ-poitiers.fr

1. INTRODUCTION

Squeeze film dampers (SFD) are components that bring an efficient damping to the rotors of modern aircraft engines. These rotors are light, flexible structures supported exclusively by ball and roller bearings that have low or almost no damping. The use of squeeze film dampers enables the design of complicated rotors operating at supercritical regimes. As shown in Fig. 1, the squeeze film damper consists of a very thin oil film interposed between the outer ring of the bearing and the engine structure. The squeeze film is similar to a lubricated bearing but none of its surface is rotating. The only mobility is the radial squeeze motion generated by the vibration of the rotor. The squeeze motion and the viscosity of the lubricating oil (which is the same that lubricates the ball bearing) will generate the damping force. Lubrication models based on the traditional Reynolds equation can describe forces inside the squeeze film but the results are seldom of practical value. There are two objective reasons for this disagreement. The first is that squeeze film dampers are always provided with feeding orifices, grooves and piston ring seals that are not always taken into account or are completely discarded in pure analytical approaches. However, these design details have a non-negligible impact on the pressure field inside the squeeze film. The second reason why traditional Lubrication models are not in agreement with reality is that operating conditions of squeeze film dampers often lead to thin film flows dominated by inertia forces and two forms of dynamic cavitation: vapor and gaseous. The traditional Reynolds equation cannot handle convective inertia effects and need very specific models for dealing with dynamic cavitation. Theoretical models aimed to overcome these difficulties were introduced in the past by the author [1-3] and his co-workers and are now synthetically presented together with comparisons with experimental results.

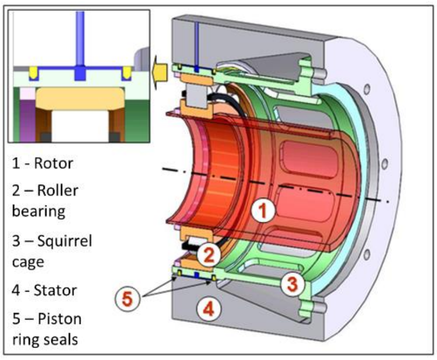


Fig. 1 Squeeze film damper layout

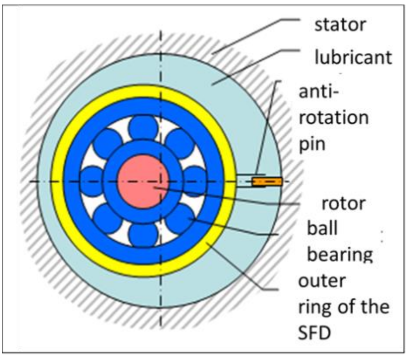


Fig. 2 Simplified SFD

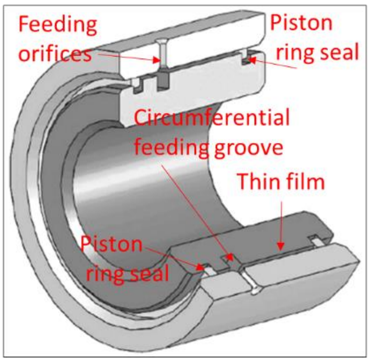


Fig. 3 Components of a real SFD

2. MATHEMATICAL MODEL

A simplified SFD is depicted in Fig. 2. The vibration of the rotor induce a squeeze motion in the thin oil film contained between the outer ring of the bearing and the casing. The “reduced” Reynolds number characterizing this flow is, $Re^*=\rho\omega C^2/\mu$, where ρ and μ are the density and the dynamic viscosity of the oil at the operating temperature, ω is the angular velocity of the whirl (squeezing) motion and C is the radial clearance, generally of the order of $1\cdot10^{-3}\ldots5\cdot10^{-3}$ of the radius or larger. If $Re^*<1$, then the use of Reynolds equation for thin film flow is appropriate. This equation discards convective inertia, is quite easy to solve and its simplified forms have analytic solutions. However, SFD of modern aircraft turbo-engines often operate at $Re^*>1$ because either ω or C have increasingly larger values. The results given then by Reynolds equation are only of qualitative nature while the design of turbo-engines require values of the dynamic stiffness and

Unsteady effects due to unsteady flow separations at low Reynolds numbers for stationary airfoils

Dan MATEESCU*

*Corresponding author
Doctor Honoris Causa, FCASI, AFAIAA, Erskine Fellow,
Professor Emeritus, Mechanical Engineering Department, McGill University,
Montreal, QC, Canada, dan.mateescu@mcgill.ca

***Abstract**: This paper presents the unsteady effects due to the flow separations on the stationary airfoils at low Reynolds numbers, discovered by the author and his graduate students, which appear starting from relatively small angles of attack, such as 6 or 8 degrees. The results are obtained using a time-accurate numerical method developed by the author and his graduate students for the solution of the Navier-Stokes equations for unsteady incompressible flows at low Reynolds numbers.*

1. INTRODUCTION

The flow past airfoils and wings at low Reynolds numbers (600 to 4000) received recently a special interest due to the development of the unmanned micro-air-vehicles (MAV), which can operate in various indoors or outdoors environments including tunnels, desert and jungle. These flows are dominated by viscous effects and flow separation phenomena, which appear at relatively small angles of attack, and are very different from the flows at regular Reynolds numbers of millions.

The steady flows past airfoils at low Reynolds numbers have been studied by Kunz & Kroo [1] and by Mateescu & Abdo [2, 3].

The unsteady flows past oscillating airfoils have been studied by Mateescu and his graduate students for airfoils executing oscillatory pitching motions [4] and in the presence of the ground [5], with a time-accurate numerical method developed by them for the solution of the Navier-Stokes equations for unsteady incompressible flows at low Reynolds numbers. They also studied the unsteady confined laminar flows [6].

The three-dimensional flows in confined configurations with oscillating walls and time-variable inflow velocities have also been studied by Mateescu & Panahi [7] with a new time-accurate method for the solution of the Navier-Stokes equations in incompressible laminar flows.

The method was in good agreement with the experimental results for steady flows, indicating for the first time the effect of the experimental lateral walls.

2. UNSTEADY NUMERICAL METHOD

A time-accurate method of second-order-accuracy has been developed for the solution of the Navier-Stokes equations of unsteady incompressible flows at low Reynolds numbers.

The problem is solved in a rectangular computational domain obtained by special transformations, in which the modified Navier-Stokes equations are discretized in real time and a pseudo-time relaxation procedure based on artificially-added compressibility is used.

A pseudo-time discretization is utilized together with a factored alternative-direction-implicit (ADI) scheme in conjunction with a spatial discretization based on a stretched staggered grid (which avoids the odd-and-even points decoupling) and sine and tangent stretching functions. A special decoupling procedure is then used to eliminate the pressure based on the continuity equation, and to reduce the problem to the solution of decoupled scalar tridiagonal systems of equations, enhancing substantially the computational efficiency.

3. METHOD VALIDATION

The method is first validated by comparison with the experimental results obtained in Japan by Prof. Asai and his students [9, 10] for the steady flow past a triangular airfoil at Reynolds number 3000.

The time-averaged values of the present solutions for the lift and drag coefficients, C_L and C_D, are compared with the experimental results in Figure 1, which includes also the maximum and minimum values

Comparison of critical behaviors of elliptic and hyperbolic quadratic algebraic equations with variable coefficients

Adriana NASTASE*

*Corresponding author
RWTH, Aachen University, Aerodynamics of Flight,
Templergraben 55, 52062 Aachen, Germany
nastase@lafaero.rwth-aachen.de

Abstract: *A comparison of the behaviors of the elliptic with those of hyperbolic quadratic algebraic equations (QAEs) with free and linear variable coefficients, in vicinity of their critical surfaces is made. The critic values of the elliptic and hyperbolic QAEs with variables coefficients are obtained by can-celling their great determinant. If only the free term of a QAE is variable from - ∞ to + ∞ and the QAE are two-dimensional, an elliptic QAE is represented by coaxial ellipses, which decrease in size and collapse in their common center. A hyperbolic QAE is represented by coaxial hyperbolas, which approach their asymptotes, degenerate in them, jump over them and go away from them. The real solutions of hyperbolic QAEs exist for all the values of free term and for elliptic QAE, if the value of the free term is greater than the critical one, the real solutions of elliptic QAEs do no longer exist. If, additionally, also the free term is variable, critical parabolas occur, if a plane of coefficients is used. The real solutions for elliptic QAE collapse along their critical parabola and do not exist inside of it. The hyperbolic QAE is represented by coaxial hyperbolas which degenerate in their asymptotes and jump over them along their critical parabola.*

Key Words: *Qualitative analysis of quadratic algebraic equations (QAEs) with variable coefficients, Collapses and holes of the real solutions of elliptic QAEs, Degeneration, jumps and breaks/rebuilds of real solutions of hyperbolic QAEs*

1. INTRODUCTION

Let us consider a QAE of elliptic or hyperbolic type namely,

$$\sum_{i=1}^{M}\left[\sum_{j=1}^{M} a_{ij}x_i x_j + 2a_{i,M+1}x_i\right] + a_{M+1,M+1} = 0. \tag{1}$$

Three determinants play an important role in the qualitative analysis it is:
- its discriminant δ,

$$\delta = \begin{vmatrix} a_{11} & a_{12} & \cdots & a_{1M} \\ a_{21} & a_{22} & \cdots & a_{2M} \\ \vdots & \vdots & & \vdots \\ a_{M,1} & a_{M,2} & \cdots & a_{M,M} \end{vmatrix}, \tag{2}$$

- its great determinant Δ:

$$\Delta = \begin{vmatrix} a_{11} & a_{12} & \cdots & a_{1,M} & a_{1,M+1} \\ a_{21} & a_{22} & \cdots & a_{2,M} & a_{2,M+1} \\ \vdots & \vdots & & \vdots & \vdots \\ a_{M,1} & a_{M,2} & \cdots & a_{M,M} & a_{M,M+1} \\ a_{M+1,1} & a_{M+1,2} & \cdots & a_{M+1,M} & a_{M+1,M+1} \end{vmatrix}, \tag{3}$$

- and its characteristic determinant Δ_c

$$\Delta_c = \begin{vmatrix} a_{11}-\lambda & a_{12} & \cdots & a_{1,M} \\ a_{21} & a_{22}-\lambda & \cdots & a_{2,M} \\ \vdots & \vdots & & \vdots \\ a_{M,1} & a_{M,2} & \cdots & a_{M,M}-\lambda \end{vmatrix}, \tag{4}$$

SECTION 1 – Aerodynamics Design

The influence of aerodynamic forces on the vehicle bodywork of railway traction

Sorin ARSENE*[,1], Ioan SEBESAN[1]

*Corresponding author
[1]"POLITEHNICA" University of Bucharest, Transport Faculty,
Depart Rolling Stock Railway,
Splaiul Independentei no. 313, Sector 6, Code 060042, Bucharest, Romania
sorinarsene@gmail.com*, ioan_sebesan@yahoo.com

Abstract: *Increase of the speed of moving in the railway system requires the development of some analysis on the vehicles aerodynamics and how it affects its performance, or efforts that appear on the various components. The aerodynamic forces have a greater impact in the case of medium and large values of the relative velocity of the air flow near the vehicle. This paper aims at an analysis of the tasks caused by the aerodynamic forces in case of the bodywork to electric locomotive, type LE 060 EA of 5100 kW. In this effect was modelled in 3D format the bodywork and the chassis of locomotive, after which they were made a series of simulations on air flow for different values of the velocity of the vehicle within the range 0 km/h – 200 km/h.*

Key Words: *bodywork structure, aerodynamic forces, railway vehicles.*

1. INTRODUCTION

At moving the engines railway vehicles, the thrust developed their acting of power equipment and the amount of resistant forces opposing the movement in the desired direction. [1]

The resistance forces acting while the vehicle is moving on the towing section profile which is in alignment and plane (straight line without ruling gradients) are determined by friction, for instance: friction on the axles bearings, rolling and / or sliding friction, road surface friction, air friction, etc. [1]

Generalized formula for determining the running resistance for railway vehicles also is known as Davis's relationship [2-5] is:

$$R_{veh} = A + B \cdot v + C \cdot v^2 \tag{1}$$

where R_{veh} – Total resistance to motion of the vehicle; A – Mechanical rolling resistances caused by the axle loads; $B \cdot v$ – Non-aerodynamic drag; $C \cdot v^2$ – Aerodynamic drag; v – Speed of the vehicle.

Davis' constants from the resistance to motion formula depend on the type and characteristics of each vehicle.

In the case of the locomotive LE 060 EA by 5100 these constants are given in the literature of the field [1, 6], and summarized in Table 1:

Table 1 – The values of Davis's constants form locomotive LE 060 EA by 5100

Type of vehicle	A [N]	B [N/(km/h)]	C [N/(km/h)2]
LE 060 EA - v.1 (120 t)	1770	5,9	0,333
LE 060 EA - v.2 (120 t)	1500	12	0,3

Explicit formula for the parameter "*c*" regarding aerodynamic resistances according to the literature [7 - 15], is:

$$c = \frac{C_x \cdot S \cdot \rho}{2} \tag{2}$$

where: $C_x = \frac{2 \cdot F_{ax}}{S \cdot \rho \cdot v^{-2}}$ – aerodynamic coefficient of air gliding, also known under the name of air penetration coefficient (dimensionless); S – the frontal area of the vehicle in the cross-sectional area (m^2); ρ – the air density in the moving vehicle (kg/m^3); F_{ax} – drag frontal force (N); v – the speed of the fluid (air) (m/s).

SECTION 2 – Numerical Analysis

Two and Three Dimensional Blood Flow Simulations in Different Types of Blood Vessels

Balazs ALBERT*

*Corresponding author
Babes-Bolyai University
No. 1 Mihail Kogalniceanu, 400084, Cluj-Napoca, Romania
balazsalb@gmail.com

Abstract: *In this paper we present a synthesis of our results obtained on blood flow simulation in different types of blood vessels. We present first some remarks on the wall shear stress (WSS) in the case of a human abdominal aortic aneurysm (AAA), then we concentrate on the mechanical conditions which would lead to the "rupture" of the vascular vessel with aneurysm and implicitly to a possible stroke.*

We also make some investigations on the Fahraeus-Lindqvist effect in arterioles. Considering an axial-symmetric reservoir full of blood and which is linked to an arteriole (with the same particular geometry), we have pointed out in this arteriole the concentration of the red blood cells towards the core of the vessel.

To improve our work we are considering a real three dimensional geometry, which is a serious jump versus our previous results, where only axial-symmetric geometries were considered. In this respect we have reconsidered the case of a carotid artery stenosis with and without a stent.

Key Words: *non-Newtonian model for blood flow, abdominal aortic aneurysm, stenosis, Fahraeus-Lindqvist effect*

1. GLOBAL CONDITION FOR THE "RUPTURE" OF THE VASCULAR VESSEL

In the sequel we will try to get a mechanical condition for predicting the rupture risk of an AAA, such aneurysm could show up in different parts and different forms of the body. This condition which doesn't involve clinical or experimental investigations is based on a mechanics of continua analysis.

Namely it is known that the cohesion forces which held together the particles of a continuum – as the aortic wall are related to the internal stress $\vec{T}$.

The analyzed rupture takes place when this internal stress of the boundary (in our case given by the general Maxwell model for viscoelasticity) is overpassed by the so called "wall shear stress" (WSS) evaluated on the considered boundary.

We will illustrate our research for the real clinical case of a human abdominal aorta with a double aneurysm, as it can be found in the paper elaborated by Finol et al. 2002

A numerical simulation for this arterial segment has been made. Evaluating the *WSS* along the vessel wall and focusing on the 5 particular points of the vessel wall we got the following distribution of the *WSS* in the case of the considered *AAA*.

It has been shown that the maximum value for *WSS* is reached in a point located between the two aneurysms and is a point located after the second one.

If the projection of the internal (vessel wall) stress $\vec{T}$ on the corresponding tangent unit vector $\vec{t}$ is overpassed *WSS* a "rupture" of the vessel wall could take place.

2. ON THE FAHRAEUS-LINDQVIST EFFECT IN ARTERIOLES

The Fahraeus-Lindqvist effect is the migration of red blood cells (RBCs) to the axial core in arterioles. This leads to the formation of high viscosity in RBC-rich core and of a low-viscosity in an RBC-free plasma layer causing a nonlinear blood flow velocity profile [16].

We note that the aria of the cell poor layer is comparable with that of the rich red blood cells in the central zone.

At low shear rates, as in arterioles – where a non-Newtonian model for blood flow should be used, the RBCs tend to aggregate and thus exhibit an increase of viscosity.

In what follows we intend to check numerically the Fahraeus-Lindqvist effect, by using an adequate mathematical model for the blood flow. We will use a blood vessel configuration made by an axial-symmetry reservoir (artery) linked to an arteriole with the same particular geometry (axial-symmetry).

Leading Edge Device Aerodynamic Optimization

Marius Gabriel COJOCARU*[,1], Mihai Leonida NICULESCU[1], Dumitru PEPELEA[1]

*Corresponding author
[1]INCAS – National Institute for Aerospace Research "Elie Carafoli",
Flow Physics Department, Numerical Simulation Unit
B-dul Iuliu Maniu 220, Bucharest 061126, Romania
cojocaru.gabriel@incas.ro*, niculescu.mihai@incas.ro, pepelea.dumitru@incas.ro

Abstract: *Leading edge devices are conventionally used as aerodynamic devices that enhance performances during landing and in some cases during takeoff. The need to increase the efficiency of the aircrafts has brought the idea of maintaining as much as possible a laminar flow over the wings. This is possible only when the leading edge of the wings is free from contamination, therefore using the leading edge devices with the additional role of shielding during takeoff. Such a device based on the Krueger flap design is aerodynamically analyzed and optimized. The optimization comprises three steps: first, the positioning of the flap such that the shielding criterion is kept, second, ,the analysis of the flap size and third, the optimization of the flap shape. The first step is subject of a gradient based optimization process of the position described by two parameters, the position along the line and the deflection angle. For the third step the Adjoint method is used to gain insight on the shape of the Krueger flap that will extend the most the stall limit. All these steps have been numerically performed using Ansys Fluent and the results are presented for the optimized shape in comparison with the baseline configuration.*

Key Words: *high lift devices, optimization, laminar wing.*

1. INTRODUCTION

A solution to increase efficiency is to reduce drag by making the flow laminar over several major components. Such solutions are Natural Laminar Flow (NLF) and Hybrid Laminar Flow Control (HLFC). Solutions that rely on laminar flow to reduce drag are greatly affected by the contamination of their leading edges during the take-off. In this work we present a method for optimizing such a leading edge device in the shape of a Krueger flap that is able to recover the aerodynamic performance of a slat. The optimization process is numerically performed in Ansys Fluent and Ansys Workbench environments. The process consists of three steps: the determination of the Krueger flap size, the optimization of the Krueger flap position and the optimization of the Krueger flap shape.

In the optimization process the observable is the *gliding ratio*, whose sensitivity is computed with reference to the shape change.

The typical use of the adjoint solver involves the following steps:

1. Compute a conventional flow solution with the above assumptions.
2. Load the adjoint solver.
3. Specify the observable of interest and set the adjoint solver controls, monitors and convergence criteria.
4. Initialize the adjoint solution and iterate to convergence.

2. OPTIMIZATION PROCESS

As stated previously the optimization consists of three major steps. In the following we will discuss in detail each individual step.

2.1 Krueger flap size

The size of the Krueger flap has to be addressed by taking into account the maximum extent of the stowed flap and the position of the wings torsion box and the space allocation in front of it. Considering that the latter is fixed and that the forward extent of the stowed Krueger cannot go below 1% of the chord and that the Krueger's upper surface must coincide with the clean airfoil lower side, we are left to optimize the Krueger size by analyzing the influence of the Krueger's trailing edge on its performance.

When using the 1% of the stowed Krueger (maximum Krueger size) we would expect to have a higher lift when compared to 1.5% and 2% (minimum Krueger size considered), since the area is smaller a lower lift is expected. When comparing the flow at AoA of 5° we can see a detached flow at the Krueger's Trailing

Flow Patterns in the Magnetic Fluid Core of a Miniature Planar Spiral Transformer

J. B. DUMITRU[1], A. M. MOREGA*[,1,2], M. MOREGA[1], L. PISLARU-DANESCU[3]

*Corresponding author
[1]"POLITEHNICA" University of Bucharest, Faculty of Electrical Engineering, Bucharest, Romania
dumitrujeanbogdan@yahoo.com, amm@iem.pub.ro*, mihaela@iem.pub.ro
[2]"Gheorghe Mihoc – Caius Iacob" Institute of Statistical Mathematics and Applied Mathematics, Bucharest, Romanian Academy
[3]National Institute for Electrical Engineering ICPE-CA, Bucharest, Romania
lucian.pislaru@icpe-ca.ro

Abstract: *Energy harvesting devices (EHD) use small-scale components with low losses. Electric power transformers (EPTs) key parts of EHDs, are utilized to convert the voltage/current power parameters from the primary, energy harvesting stage levels to the secondary, storage and delivery stage levels. Magnetic colloidal nanofluids (MNF) seem to be a sound solution for the magnetic circuit, enabling miniaturized constructions, whose implementation may benefit of LIGA fabrication technology. The occurring magnetization body forces within the magnetic fluid core may be complex and have to be evaluated, and this paper is concerned with their study. Several common powering schemes for the MPST are envisaged and the pending flows investigated.*

Key Words: *Transformer; spiral, planar windings; flow; numerical simulation; finite element; magnetic colloidal nanofluid.*

1. INTRODUCTION

A key component in *Energy Harvesting Systems* (EHS) is the power transformer of the fly-back convertor, which must comply to certain specifications such as small size, low profile, thermal stability, high efficiency, and low cost. *Micro-Electromechanical Systems* (MEMS) *Miniature Planar Spiral Transformers* (MPST) are sound solutions, and their study and development is of current concern [1-7]. Their magnetic core is usually made of ferrite, but recent studies show that *magnetic nanofluids* (MNF) may provide for higher power conversion efficiency [5-15], because they may alleviate the difficulties related to the magnetic core losses, especially at high frequencies, due to their almost zero hysteresis. This study is concerned with a MPST concept that utilizes MNF as part of its magnetic core. Electric and magnetic field problems are solved to analyze the forced, magnetically produced flow within the fluid part of the MNF core. The results indicate that the usage of 2D rather than 3D models may be of concern, at least in what regards the flow of the MNF core.

2. THE MINIATURE PLANAR SPIRAL TRANSFORMER

The MPST in this study has two circular copper windings, 20/40 turns, which are "grown" in LIGA [16] technology on a ceramic substrate (Al_2O_3). The magnetic casing and column are made of 3F3 ferrite, while the spacing between the windings is filled with MNF (Fig. 1).

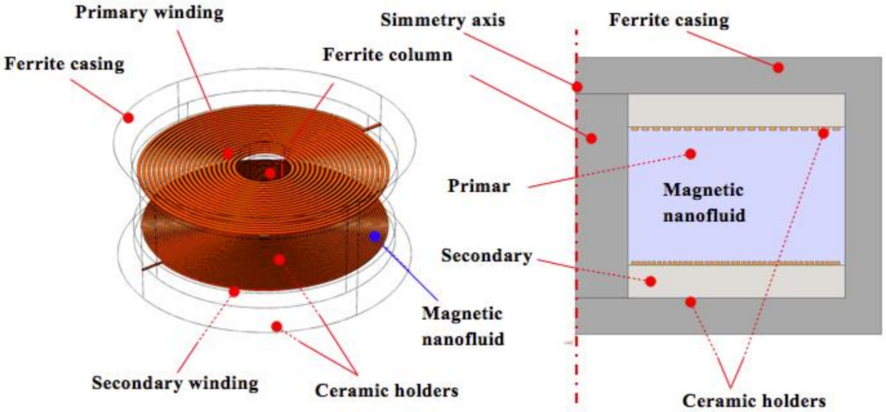


Fig. 1 The MPST with ferrite – MNF core, and the equivalent, simplified, 2D axial model [17]

Computations of the Axisymmetric Flow with Shock Waves at Hypersonic Velocities Taken into Account the Chemical Reactions that Appear in the Air at High Temperatures

Mihai Leonida NICULESCU*,[1], Marius Gabriel COJOCARU[1], Mihai Victor PRICOP[1]
Dumitru PEPELEA[1], Mihaita Gilbert STOICAN[1], Maria Cristina FADGYAS[1]

*Corresponding author
[1]INCAS − National Institute for Aerospace Research “Elie Carafoli”,
Flow Physics Department, Numerical Simulation Unit,
B-dul Iuliu Maniu220, Bucharest 061126, Romania
niculescu.mihai@incas.ro*, cojocaru.gabriel@incas.ro, pricop.victor@incas.ro,
pepelea.dumitru@incas.ro, stoican.gilbert@incas.ro, breaban.maria@incas.ro

Abstract: *The temperature in the inlet of ramjet and scramjet vehicles is extremely high due to the normal and oblique shock waves that appear at hypersonic velocities. In this case, the assumption of a calorically perfect non-reacting gas with the ratio of specific heats $\gamma = 1.4$ gives unrealistically high values of temperature. Furthermore, the classic Taylor-Maccoll equations that describe the axisymmetric shock waves are not valid in this case. For this reason, the present paper proposes an alternative to the classic Taylor-Maccoll equations, which takes into account the chemical reactions (dissociation and ionization) that appear in the air at high temperatures.*

Key Words*: Hypersonic flow, asymmetric shock waves, chemical reactions, CFD.*

1. INTRODUCTION

It is well known that at high temperatures, the air become to dissociate and ionize. For example, for air at one atmosphere, the molecular oxygen becomes to dissociate at about 2500 K and it is essentially totally dissociated at 4000 K.

At this temperature, molecular nitrogen begins and it is essentially totally dissociated at 9000 K. Above this temperature, ions are formed and the gas becomes a partially ionized plasma [1].

Unfortunately, the kinetics of these chemical reactions (cr) is quite, not very well understood and requires huge computational resources.

For example, only the air dissociation requires at least 5 chemical reactions with third body efficiency and five species (O_2, O, N_2, N, NO) [2, 3].

For this reason, the present paper presents a simplified CFD method for inviscid axisymmetric flows, which takes into account both the released or absorbed heat Q due to the chemical reactions and the modification of global gas constant R due to the chemical reactions and compressibility effects.

2. GOVERNING EQUATIONS FOR INVISCID AXYSIMMETRIC SHOCK WAVES WITH CHEMICAL REACTIONS

The axisymmetric shock wave for the gases with constant ratio of specific heat is given by Taylor-Maccoll equation:

$$\begin{aligned}&\frac{\gamma-1}{2}\left[V_{\max}^2-V_r^2-\left(\frac{\mathrm{d}V_r}{\mathrm{d}\theta}\right)^2\right]\left(2V_r+\frac{\mathrm{d}V_r}{\mathrm{d}\theta}\cot\theta+\frac{\mathrm{d}^2V_r}{\mathrm{d}\theta^2}\right)-\\&\frac{\mathrm{d}V_r}{\mathrm{d}\theta}\left(V_r\frac{\mathrm{d}V_r}{\mathrm{d}\theta}+\frac{\mathrm{d}V_r}{\mathrm{d}\theta}\frac{\mathrm{d}^2V_r}{\mathrm{d}\theta^2}\right)=0\end{aligned} \tag{1}$$

In the flows with chemical reactions, the ratio of specific heat γ is not constant anymore. Moreover, one prefers another definition of γ instead of the classic one using the law of ideal gases [4]:

Supersonic flow over bidimensional model

Dumitru PEPELEA*,[1], Florin FRUNZULICA[2], Marius Gabriel COJOCARU[1]

*Corresponding author
[1]INCAS – National Institute for Aerospace Research “Elie Carafoli”
Flow Physics Department, Numerical Simulation Unit,
B-dul Iuliu Maniu 220, Bucharest 061136, Romania
pepelea.dumitru@incas.ro*, cojocaru.gabriel@incas.ro
[2]“POLITEHNICA” University of Bucharest, Faculty of Aerospace Engineering,
Str. Gheorghe Polizu nr. 1, Bucharest 0011061, Romania
ffrunzi@yahoo.com

Abstract: *This paper presents an investigation of the flow through a supersonic wind tunnel, which is practically a Laval nozzle. In first approximation, the parameters of the flow were calculated using the shockwave equations and relations describing the mass flow rate through the nozzle. 2D and 3D numerical simulations were also conducted in Ansys Fluent in order to analyze the supersonic flow over a wedge, starting from parameters provided by analytical solutions. The objective is to compare the pressure field and Mach number from the CFD simulation with available experimental data. Another objective is the visualisation of the shockwaves and supersonic expansion. Comparison between experimental data and CFD results is presented and discussed.*

Key Words: *Laval nozzle, CFD, wedge, wind tunnel.*

1. INTRODUCTION

This paper presents the study of the flow through Laval nozzle over a wedge:

- Theoretical approach: the parameters of the flow were obtained by using the equations of the normal shock waves and mass flow rate.
- Numerical simulation: the flow problem is solved with a CFD program, in this case ANSYS FLUENT.
- Experimental data: in order to visualize the flow field and measure pressures at certain cross-sections an experiment was conducted.

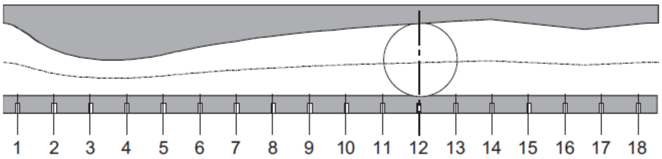


Fig. 1 – Configuration of the supersonic wind tunnel for *M = 1.8* and measuring points position

2. GOVERNING EQUATIONS FOR NORMAL SHOCK WAVES AND MASS FLOW RATE

Using the conservation laws of continuity, momentum and energy we can write the equations of the normal shock wave[1]:

$$\frac{p_2}{p_1} = \frac{2\gamma M_1^2 - (\gamma - 1)}{\gamma + 1} \tag{1}$$

$$\frac{\rho_2}{\rho_1} = \frac{(\gamma + 1)M_1^2}{(\gamma - 1)M_1^2 + 2} \tag{2}$$

Another relation used to calculate the mass flow rate is Saint-Venant equation :

$$\dot{m} = \sigma \sqrt{\frac{2\gamma}{\gamma - 1} p_1 \rho_1 \left(\frac{p}{p_1}\right)^{\frac{2}{\gamma}} \left[1 - \left(\frac{p}{p_1}\right)^{\frac{\gamma - 1}{\gamma}}\right]} \tag{3}$$

SECTION 3 – Equations of Mathematical Physics

The flow of an Oldroyd-B fluid between two parallel plates with a porous top plate

Corina CIPU*,[1], Victor TIGOIU[2]

*Corresponding author
*,[1]UPB – POLITEHNICA University Bucharest, Faculty of Applied Sciences,
Splaiul Independenţei 313, 060042, Bucharest, Romania
corina_cipu@mathem.pub.ro
[2]Faculty of Mathematics and Computer Science, University of Bucharest,
Academiei 14, Bucharest 010014, Romania
vtigoiu@fmi.unibuc.ro

Abstract: *In this paper we analyze if von Kármán type solutions are admissible for an Oldroyd-B fluid flow problem between two parallel plates. The study of the corresponding boundary value problem is performed in order to obtain analytical solutions. The nonlinear problem involved is analyzed by means of a double scale asymptotic development. For this purpose we develop an homotopy analysis method (HAM) and an iterative method, respectively. Finally the theoretical results are plotted in some graphs in order to point out the character of the fluid flow under consideration.*

Key Words: *non-newtonian fluids, Oldroyd-B fluid, homotopy analysis method.*

1. INTRODUCTION

For a third grade fluid the problem was studied by Tigoiu in [2] and [3]. A numerical approach of this problem, for some constitutive parameters is described by Cipu, Pricina and Tigoiu in [4]. The nonlinear thermal problem was also solved numerically by Cipu and Pricina ([5]). In this paper we consider a fluid flow between two parallel plates at a distance "d" one to another.

The upper plate is porous and fixed. The fluid is going down throughout the upper plate with constant vertical velocity v_0: $\boldsymbol{v}(x,y)\big|_{y=d} = -v_0\boldsymbol{e}_2$, taking place the fluid mass conservation between plates. The lower plate is moving with the velocity: $\boldsymbol{v}(x,y)\big|_{y=0} = -cx\boldsymbol{e}_1$, $\boldsymbol{e}_1, \boldsymbol{e}_2$ being versors in the horizontal and vertical directions, and "c" a given constant. The origin of the absolute referential system is supposed to be fix. A solution of *von Kármán* type is given by:

$$u = cxf'(\eta)\,,\ v = -cdf(\eta)\,, \text{ where } \boldsymbol{v}(x,y) = u\boldsymbol{e}_1 + v\boldsymbol{e}_2\,,\ y = d\eta\,. \tag{1}$$

The incompressible fluid has a plane flow. The stress is supposed to be independent of z, and is given by the constitutive equation for an Oldroyd-B fluid (see Oldroyd [1]) i.e.:

$$\mathbf{T} = -p\mathbf{I} + \mathbf{T}_E\,,\ \mathbf{T}_E + \lambda\frac{D\mathbf{T}_E}{Dt} = \mu(\mathbf{A}_1 + \lambda_1\frac{D\mathbf{A}_1}{Dt})\,. \tag{2}$$

Here $\mathbf{T}$ is the Cauchy stress tensor, $\mathbf{T}_E$ is the effective stress, $\mathbf{A}_1 = \mathbf{L} + \mathbf{L}^{\mathbf{T}}$ is the first Rivlin-Ericksen tensor, $p(\boldsymbol{x},t)$ is the hydrostatic pressure, μ is the dynamical viscosity (supposed to be constant), and λ, λ_1 are relaxation times.

The convective derivatives from (2) have the following form:

$$\frac{D\mathbf{T}_E}{Dt} = \dot{\mathbf{T}}_E + \mathbf{T}_E\mathbf{L} + \mathbf{L}^{\mathbf{t}}\mathbf{T}_E\,, \frac{D\mathbf{A}_1}{Dt} = \dot{\mathbf{A}}_1 + \mathbf{A}_1\mathbf{L} + \mathbf{L}^{\mathbf{t}}\mathbf{A}_1\,. \tag{3}$$

We investigate if von Kármán's type solutions are admissible for an Oldroyd-B fluid subject to the flow problem described above.

In the absence of mass forces the equations of motion are reduced to $\rho\boldsymbol{a} = \mathrm{div}\mathbf{T}$ which, after long but straightforward calculi, lead to:

$$\mathrm{Re}\,\overline{x}(f'^2 - ff'') = \frac{\partial\overline{\mathrm{T}}_E^{\,11}}{\partial\overline{x}} + \frac{\partial\overline{\mathrm{T}}_E^{\,12}}{\partial\eta} - \frac{\partial\overline{p}}{\partial\overline{x}}\,,\ \mathrm{Re}\,ff' = \frac{\partial\overline{\mathrm{T}}_E^{\,12}}{\partial\overline{x}} + \frac{\partial\overline{\mathrm{T}}_E^{\,22}}{\partial\eta} - \frac{\partial\overline{p}}{\partial\eta}\,, \frac{\partial\overline{\mathrm{T}}_E^{\,13}}{\partial\overline{x}} + \frac{\partial\overline{\mathrm{T}}_E^{\,23}}{\partial\eta} = 0\,. \tag{4}$$

Model reduction by moment matching for boundary control PDEs

Tudor Corneliu IONESCU*[,1], Orest Vasile IFTIMIE[2], Alexandru DUMITRACHE[1]

*Corresponding author
[1]"Gheorghe Mihoc-Caius Iacob" Institute for Mathematical Statistics and Applied Mathematics of the Romanian Academy,
Calea 13 Septembrie 13, 050711 Bucharest, Romania
td.c.ionescu@gmail.com*, alex_dumitrache@yaho.com
[2]University of Groningen, Faculty of Economics and Business,
Nettelbosje 2, 9747 AE, Groningen, The Netherlands
o.v.iftime@rug.nl

The tendency to design and analyse systems of increasing complexity has received a growing interest during the last years. Along with this tendency, the complexity of the mathematical models increases both in nature and dimension.

Complex models are more difficult to analyse, and it is also more difficult to develop operating and/or open/closed-loop control algorithms in a systematic way.

Therefore, model reduction is of utmost importance to obtain models, useful for analysis, simulation and control.

The model reduction technique we use is the so-called time-domain moment matching method [1]. The idea is that the steady-state response of the approximant when excited by some specific input signal matches the steady-state response of the given model, when excited by the same input.

The result of applying this method is a family of parametrized approximations which contains a wide variety of models suitable for different goals.

We are going to explore and exploit this asset for a particular fractional system [2, 3], namely a perturbed wave equation.

Consider the linear infinite dimensional system

$$\partial_t^2 W(t,x) + 2\varepsilon\partial_t^{\frac{3}{2}} W(t,x) + \varepsilon^2\partial_t^1 W(t,x) - \partial_x^2 W(t,x) = 0, \tag{1}$$

where t > 0 and $x \in (0,1)$, with the initial conditions

$$W(0,x) = 0, \text{ and}$$

$$\partial_x W(t,x) = 0,\ x \in (0,1).$$

The state $W(t,x)$ is infinite dimensional, and $\varepsilon > 0$ is a small perturbation parameter of the wave PDE. Taking viscothermal losses into account in the propagation of the air in the cylinder leads to the modified wave PDE.

The (dynamic) boundary conditions are taken to be

$$\begin{aligned} &\left[a_0\left(\partial_t + \varepsilon\partial_t^{\frac{1}{2}}\right) + b_0\partial_{-x}\right] W(t,0) = a_0\left(\partial_t + \varepsilon\partial_t^{\frac{1}{2}}\right)u(t), \\ &\left[a_0\left(\partial_t + \varepsilon\partial_t^{\frac{1}{2}}\right) + b_1\partial_x\right] W(t,0) = 0,\ t > 0, \end{aligned} \tag{2}$$

where $a_0 b_0 > 0,\ a_1 b_1 > 0$, and $u(t), t > 0,$ is the input of the system.

The output of the perturbed wave is

$$y(t) = W(t,1). \tag{3}$$

The transfer function of the system defined by the equations ((1),(2),(3)) is

$$G(s) = \frac{e^{-(s+\varepsilon\sqrt{s})}}{1-\rho e^{-2(s+2\varepsilon\sqrt{s})}}, \tag{4}$$

Continuous defects: dislocations and disclinations in finite elasto-plasticity with initial dislocatons heterogeneities

Raisa PASCAN*,1, Sanda CLEJA-TIGOIU2

*Corresponding author
*,1Faculty of Applied Sciences, "POLITEHNICA" University of Bucharest,
Splaiul Independenţei 313, 060042, Bucharest, Romania
pascanraisa@yahoo.com
2Faculty of Mathematics and Computer Science, University of Bucharest,
Academiei 14, 010014, Bucharest, Romania
tigoiu@fmi.unibuc.ro

Abstract: *In the elasto-plastic framework with finite deformation, we described a model with microstructural defects. The defect densities are related to the incompatibility of the plastic distortion,* $\mathbf{F}^p$*, and the incompatibility of the plastic connection, which involves the disclination tensor,* $\mathbf{\Lambda}$*, and has a non zero curvature. The defect densities characterize the Burgers and Frank vectors and are reduced to* $\boldsymbol{\alpha}=\mathrm{curl}\mathbf{H}^p$ *and* $\boldsymbol{\theta}=\mathrm{curl}\mathbf{\Lambda}$ *in the case of small distortions. The finite element method is applied to study numerically the evolution of the defects, when the initial dislocation heterogeneities are considered.*

Key Words*: Burgers vector; dislocation density; disclination density; dipole of disclinations; diffusion-like evolution for plastic distortion; FEM and update algorithm.*

1. INTRODUCTION

In the present model the tensorial densities of dislocations and disclinations were involved. We consider the constitutive framework developed in the paper [2], where both type of the defects: dislocations and disclinations are considered and the free energy density is formulated like in [1]. The reduced dissipation inequality allows us to derive the appropriate non-local evolution diffusion-like equations for the plastic distortion and for the disclination tensor. In the present paper we considered the model of edge dislocations coupled with wedge disclinations in the case of small elasto-plastic distortions. We numerically solved the initial and boundary value problem, when the initial heterogeneity of the defects was prescribed by the dislocation tensor able to induce in the process the dipole of disclinations.

2. WEDGE DISCLINATIONS AND EDGE DISLOCATIONS

We recall that a wedge disclination is developed within the body if $(\mathrm{curl}\mathbf{\Lambda})\mathbf{e}_3 \parallel \mathbf{e}_3$. We consider that $\Lambda_{3s} = \Lambda_{3s}(x^1, x^2),\ s = 1, 2$ are non-vanishing only. The edge dislocation, which generates a Burgers vector in the plane, is characterized by the plastic components H_{ij}^p with $i, j = 1, 2,$ being function of $(x^1, x^2),$ and

$$\mathrm{curl}\mathbf{H}^p = \left(H_{12,1}^p - H_{11,2}^p\right)\mathbf{e}_1 \otimes \mathbf{e}_3 + \left(H_{22,1}^p - H_{21,2}^p\right)\mathbf{e}_2 \otimes \mathbf{e}_3. \tag{1}$$

The evolution equations for the plastic distortion and the disclination tensor were derived in [2] and take the following form:

Proposition The fields $\mathbf{H}^p$ and $\mathbf{\Lambda}$ are described by the following evolution equations

$$\begin{aligned}\xi_1\dot{\mathbf{H}}^p &= -\beta_2\ \mathrm{curl}(\ \mathrm{curl}\mathbf{H}^p) + \beta_2 curl(\mathbf{\Lambda}^T) + \mathbf{T} + \\ &\quad + 4\beta_2\ \mathbf{\Lambda}^T\mathbf{\Lambda} - 2\beta_2\left(\mathbf{\Lambda}^T\ (curl\mathbf{H}^p)^T + (\ curl\mathbf{H}^p)\mathbf{\Lambda}\right) \\ \xi_2\ \dot{\mathbf{\Lambda}} &= \beta_4\Delta\mathbf{\Lambda} - (\beta_3 + 2\beta_2)\mathbf{\Lambda} + 2\beta_2\ (curl\mathbf{H}^p)^T - \beta_4\frac{\partial\Lambda_{3s}}{\partial x^k}\frac{\partial H_{qs}^{(p)}}{\partial x^k}\mathbf{e}_3 \otimes \mathbf{e}_q - \\ &\quad - \beta_4\frac{\partial\Lambda_{3q}}{\partial x^k}\frac{\partial(\mathrm{tr}\mathbf{H}^p)}{\partial x^k}\mathbf{e}_3 \otimes \mathbf{e}_q,\quad \forall\ s, q, k \in \{1, 2\}\end{aligned} \tag{2}$$

PROBLEM. Solve the quasi-static, initial and boundary value problems associated to the equilibrium equation $div\left(\mathcal{E}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^p)\right) = 0$ and coupled with the flow rules (2). The variational problem, which defines the

SECTION 4 – Mathematical Modeling

Issues on Modeling and Simulation of a Mixed Flow Turbofan

Irina Carmen ANDREI*[,1], Alexandra STANESCU[2]

*Corresponding author
*[,1]INCAS − National Institute for Aerospace Research “Elie Carafoli”,
Flow Physics Department, Numerical Simulation Unit,
B-dul Iuliu Maniu 220, Bucharest 061126, Romania
andrei.irina@incas.ro
[2]Universite Toulouse III - Paul Sabatier,
M2P Management International du Transport Aérien et du Tourisme,
118 Rue Narbonne, 31400, Toulouse, France
al3xandrastanescu@yahoo.com

Abstract: *The intent of this paper is to do an accurate prediction for the off-design performances of a mixed flow turbofan. Following the completion of the mathematical model of the engine, by the identification of the missing parameters, the study is focused on model setup and the determination of engine performances (thrust, specific thrust and specific fuel consumption) with the variation of flight conditions, and its altitude and flight velocity maps. The components of modular modeling approach have been integrated into a global engine model, with the intent of reducing the overall simulation effort and/ or time. In Simulink have been done dynamic simulations for the global turbofan engine.*

Key Words*: mixed flow turbofan, engine parameters, performances, off-design regimes, operating maps, numerical simulation.*

1. INTRODUCTION

In simulation of mixed flow turbofan engine operation, as well as in performance calculation at design and off-design regimes, [2-4], the mathematical model is more complex with respect to the turbojet engine, due to the presence of the bypass airflow and the mixing of it with the core gas flow. For these reasons, the aero-thermodynamic analysis of a mixed flow turbofan engine requires several stages of comprehensive study. Also, the identification of the missing engine design parameters, [1], must complete the mathematical model.

2. BASIC EQUATIONS FOR THE MIXD FLOWS TURBOFAN ENGINE

The performances of the mixed flow turbofan are the thrust F [N] (1), specific thrust F_{sp} [Ns/kg] (2) and specific fuel consumption C_{sp} [kg/Nh] (3), expressed by the relations given below:

$$F = F_{sp} \cdot \dot{M}_{a1} \tag{1}$$

$$F_{sp} = (1+K) \cdot C_{5\_am} - V \tag{2}$$

$$C_{sp} = \frac{3600}{F_{sp}} \cdot \frac{\left(i_3^* - i_2^*\right)}{\left(P_{ci} \cdot \xi_{ca} - i_3^*\right)} \tag{3}$$

The variation of altitude H[km], flight velocity V[m/s] or flight Mach number and engine operation regime ($\bar{n} = \%\, n_{design}$, or rotational speed n [rpm]), largely influences the following parameters: compressor pressure ratio or π_c^* (4), fan pressure ratio π_v^* (5), core air flow $\dot{M}_{a1}$ [kg/s] (6), bypass ratio K (7); the reference values are calculated for the SLS, ISA conditions.

$$\pi_c^* = \left(1 + \bar{n}^2 \cdot \left(\frac{i_0}{i_1^*}\right) \cdot \left(\frac{\eta_c}{\eta_{c0}}\right) \cdot \left[\left(\pi_{c0}^*\right)^{\left(\frac{k-1}{k}\right)} - 1\right]\right)^{\left(\frac{k}{k-1}\right)} \tag{4}$$

$$\pi_v^* = \left(1 + \bar{n}^2 \cdot \left(\frac{i_0}{i_1^*}\right) \cdot \left(\frac{\eta_v}{\eta_{v0}}\right) \cdot \left[\left(\pi_{v0}^*\right)^{\left(\frac{k-1}{k}\right)} - 1\right]\right)^{\left(\frac{k}{k-1}\right)} \tag{5}$$

Preliminary Design of o LSA Aircraft Using Wind Tunnel Tests

Norbert ANGI*,[1], Angel HUMINIC[1]

*Corresponding author
[1]Aerodynamics Laboratory, Transilvania University of Brasov,
29 Bulevardul Eroilor, Brasov 500036, Romania
norbert.angi@student.unitbv.ro*, angel.h@unitbv.ro

Abstract: *This paper presents preliminary results concerning the design and aerodynamic calculations of a light sport aircraft (LSA). These were performed for a new lightweight, low cost, low fuel consumption and long-range aircraft. The design process was based on specific software tools as Advanced Aircraft Analysis (AAA), XFlr 5 aerodynamic and dynamic stability analysis, and Catia design, according to CS-LSA requirements. The calculations were accomplished by a series of tests performed in the wind tunnel in order to assess experimentally the aerodynamic characteristics of the airplane.*

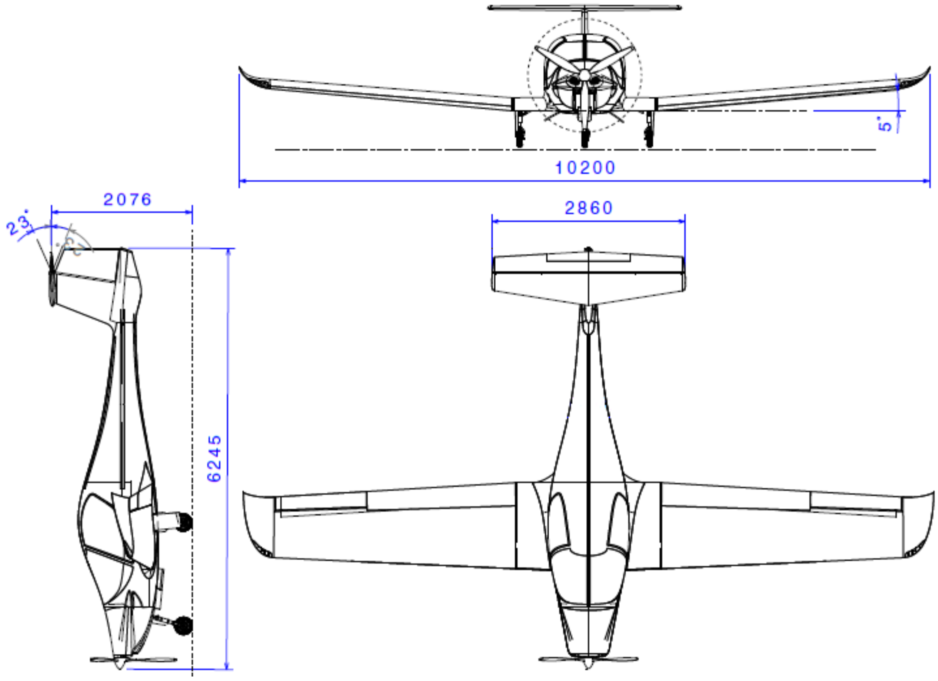
Key Words: *aircraft design, Computer Aided Design, aerodynamics, wind tunnel.*

1. INTRODUCTION

In the first stage, the aircraft structure was defined as concept, without precise calculations. The initial parameters, which were established in this step, were optimised during next stages of the design, and they were also used as input data to the 3D modelling of the aircraft, the base shape being one of the main results of the preliminary design.

The preliminary design was made with the aid of AAA [1] and XFLr 5 [2] softwares. The airplane, named *Sky Dreamer* [3], has the general characteristics shown in Table 1.

Table 1 - General characteristics of the airplane

<table>
<tr><td rowspan="10">
</td><td>Parameter</td><td>Value</td></tr>
<tr><td>Crew</td><td>Two</td></tr>
<tr><td>Empty weight
Max. Takeoff</td><td>315 kg
600 kg</td></tr>
<tr><td>Maximum speed
Cruising speed</td><td>290 km
205 km/h</td></tr>
<tr><td>Operational Range</td><td>1650 km</td></tr>
<tr><td>Service ceiling</td><td>5500 m</td></tr>
<tr><td>Lenght</td><td>6.45 m</td></tr>
<tr><td>Wing Area</td><td>11.5 m2</td></tr>
<tr><td>Wingspan</td><td>10.2 m</td></tr>
<tr><td>Wing Airfoil</td><td>Eppler 562</td></tr>
<tr><td>Fig.1 - Top, Front and Lateral views of the airplane</td><td>Fuel capacity</td><td>100 L</td></tr>
</table>

In the first stage, the aerodynamic characteristics of the airplane were assessed for the following conditions: cruise flight with gear up at 3000 m in the following atmospheric conditions: temperature $t = -4.5\,°C$, pressure $p = 70100\,N/m^2$, density $\rho = 0.909\,kg/m^3$ and dynamic viscosity $\mu = 1.69 \cdot 10^{-5}\,kg/m\,s$.

The characteristic Reynolds number computed with the mean aerodynamic chord $AMC = 1.16\,m$ was $Re = 3.81 \cdot 10^6$.

Sammon mapping for preliminary analysis in Hyperspectral Imagery

Nicolae APOSTOLESCU*[,1], Daniela BARAN[2]

*Corresponding author
*[,1]AEROSPACE Consulting,
B-dul Iuliu Maniu 220, Bucharest 061126, Romania,
apostol@incas.ro
[2]INCAS – National Institute for Aerospace Research "Elie Carafoli",
B-dul Iuliu Maniu 220, Bucharest 061126, Romania
baran.daniela@incas.ro

Abstract: *The main goal of this paper is to present the implementation of the Sammon algorithm developed for finding N points in a lower m-dimensional subspace, where the original points are from a high n-dimensional space. This mapping is done so interpoints Euclidian distances in m-space correspond to the distances measured in the n-dimensional space. This method known as non-linear projection methods or multidimensional scaling (MDS) aim to preserve global properties of points. The method is based on the idea of transforming the original, n-dimensional input space into a reduced, m-dimensional one, where m<n, and it may be used to clustering hyperspectral data. For the starting may be applied as a pre-processing procedure, Principal Component Analysis (PCA) to obtain the N points in the lower subspace. The algorithm was tested on hyperspectral data with spectra of various lengths. Depending of the size of the input data (number of points), the number of learning iterations and computational facilities available, Sammon's mapping might be computationally expensive*

Key Words: *hyperspectral data, Sammon mapping method, Euclidian distances,* non-linear mapping techniques

Hyperspectral sensors used in Hyperspectral Imagery co information of earth surfaces as a set of images that corespond to the same spatial scene, but are acquired at many different spectral bands with high resolution. These images contain abundant spatial, spectral, and radiometric information, which makes earth observation and information acquisition much more efficient for real life applications. Hyperspectral data have detail and accuracy that permit investigation of phenomena and concepts that greatly extend the scope of traditional remote sensing In terms of spectral properties, the high resolution has the capability of uncovering unknown sources, which cannot be identified by visual inspection. The spectral signature lead to better separation between physiscal materials and objects. This is based on the fact that in the hyperspectral image, reflectance information depends only of the materials spectral responses in the scene.

A mixed pixel is either linear or nonlinear combination of pure pixels signatures weigthted by the correspondent abundance fraction. Many techniques of unmixing in hyperspectral image analysis are based on geometric approach where each pixel is seen as a spectral vector of p (number of spectral bands). Under linear model assuming that the number of substances and their spectra are known, but in reality these are not known and, then, hyperspectral unmixing falls into the blindly classes. When the mixture between materials is macroscopic, the linear mixing model of spectra is generally admitted because this model assumes no interaction between materials.

The large dimensionality of a hyperspectral dataset often requires data transformation which can effectively reduce dimension of data sets with minimum loss of information. These are intended to find the minimum number of parameters required to represent the observed properties of the data. Several methods have been implemented for determining of the dimension of signal subspace that is done by the smallest number of parameters needed to contain all of the variability in the data.

This means a dimensionality reduction. The entire subject is based around the idea that we have this big set of data, and we want to analyse that set in terms of the relationships between the individual points in that data set. Hyperspectral images are characterized by a high number of bands, which are highly correlated for neighboring bands. Sammon mapping is one of the first and most popular nonlinear dimensionality reduction techniques. Also it has been widely applied to the visualization of the high dimensional data. The Sammon mapping is based on the idea of transforming the original, n-dimensional input space into a reduced, m-dimensional one, where m<n. It is also known as non-linear projection method or multidimensional scaling (MDS) method. Through this mapping function we give the minimum embedding dimension from hyperspectral data. It may be applied as a pre-processing procedure, and its resulting components may be used as inputs to clustering and supervised classification models. Sammon method place data items in an

Boolean Algebra Application in Analysis of Flight Accidents

Casandra Venera BALAN (PIETREANU)*

*Corresponding author
"POLITEHNICA" of University Bucharest, Aerospace Engineering Department
Polizu Street 1-7, sector 1, Bucharest 011061, Romania
casandra.pietreanu@yahoo.com

Abstract: *Fault tree analysis is a deductive approach for resolving an undesired event into its causes, identifying the causes of a failure and providing a framework for a qualitative and quantitative evaluation of the top event. An alternative approach to fault tree analysis methods calculus goes to logical expressions and it is based on a graphical representation of the data structure for a logic - based binary decision diagram representation. In this analysis, such sites will be reduced to a minimal size and arranged in the sense that the variables appear in the same order in each path. An event can be defined as a statement that can be true or false. Therefore, Boolean algebra rules allow restructuring of a Fault Tree into one equivalent to it, but simpler.*

Key Words: *Boolean algebra, fault tree analysis, accident investigation, probabilistic risk assessment.*

1. INTRODUCTION

As accident investigation evolves, due to the fact that the causal factors of accidents are complex, they must be broke down into categories.

The characteristics of an accident and the link between causes and effects are defined by accident models.

The induction methods answer the "What happens if…?" question and assume the existence of a component/ condition and analyze the condition's effect, while the deductive models are able to resolve the causes for an event by deducing them.

An example of a deductive method is the Fault Tree Analysis.

Fault Tree Analysis (FTA) relies on a cause-effect chain, and was developed to link failures and determine the top event (a specified undesired event) [7].

2. FAULT TREE ANALYSIS

The scenarios of failures of events in a complex system, are modeled with logical and probabilistic tools. Reliability and safety analysis are used to define, quantify and describe failures.

The length of time a component is exposed to failure, called ET (exposure time), P= 1.0 – $e^{-\lambda T}$ [6], can be controlled by testing, monitoring, etc, has a large effect on the probability calculations used in Fault Tree.

The Fault Tree is a linear method mainly used in failure laws for reliability studies, particularly applied for probability quantification and analysis of risk and maintenance tests.

FT provides a method for the cause-effect relationships and is composed of interlinked symbols. FT can break down an accident into root causes.

For example, the crash of an aircraft with loss of lives can be a top event suitable for a fault tree analysis.

Starting from the top event, fault tree determines previous faults by creating an arborescent structure on different levels.

The top event has to be properly defined in order to establish a good result and conclusion. By discovering its basic causes, the top event can be resolved.

TE is usually a system failure and it answers the questions "What?" and "When?", describing the event and the time when it happened.

Establishing the limit and the level of details on a fault tree is given by the failure data existence or the results required, knowing that at the bottom of the FT are the basic events [5].

An event can be resolved into specific causes by using the "OR" gate. This gate creates the union of the inputs; this way, the output occurs if one or more of the inputs occur.

On the other hand, the intersection of the inputs is represented by the "AND" gate.

A Hydro-Dynamical Model for Gravity

Corneliu BERBENTE*

*Corresponding author
"POLITEHNICA" University of Bucharest, Faculty of Aerospace Engineering,
Polizu no.1-6, RO-011061, Bucharest, Romania
berbente@yahoo.com

***Abstract**: A hydro-dynamical model for gravity by using an analogy with the attraction of spherical sources in incompressible fluids is proposed. Information regarding a photon-like particle called graviton is taken using an author's previous paper [5]. The substance and radiation interaction due to emission of gravitons takes place via an energy field made of the emitted gravitons and filling all the universe. The energy distribution is considered uniform at the universe scale. A consequence of the proposed model is the increasing of the universal "constant" of gravity, as a function of the age of universe.*

***Key Words:** graviton emission, "fluid of gravitons, age of universe, energy formulation*

1. INTRODUCTION

There are many theories of gravity [1-3]. Among them the general relativity which gives an interpretation on geometrical reasons, but does not use particles as carriers of interaction, unlike quantum mechanics.

In the following one tries to combine the existence of a particle called as usually graviton, but acting by means of an intermediate field also made of gravitons. This is possible by using a hydro-dynamical analogy.

2. MODEL PRESENTATION

If Q_1, Q_2 are the volume rates of two sources of incompressible fluid located in two points the hydro-dynamical force of interaction, F_H , is oriented along the direction which connects the two sources and has the expression [4]:

$$F_H = \frac{\rho Q_1 Q_2}{4\pi R_{12}^2}; \rho = const., \tag{1}$$

where ρ is the mass density of the fluid which fills the space and R_{12} the distance between sources. On the other hand, the Newton law of the universal attraction of two bodies is:

$$F_N = f_N \frac{m_1 m_2}{R_{12}^2}\ , \tag{2}$$

where m_1, m_2 are the body masses and $f_N = 6.67E-11 m^3/kg/\sec^2$ is the Newton constant of universal attraction.

One introduces now the total energy, *E*, using the special relativity formulas [8]

The Newton formula then becomes:

$$F_N = f_{NE}(t_u)\frac{E_1 E_2}{R_{12}^2}; f_{NE}(t_u) = f_N(t_u)/c^4, \tag{3}$$

$f_{NE}(t_u)$ being the new coefficient (not constant!) of universal attraction in the formulation considering energies instead of masses. This coefficient is a function of the age of universe, t_u, considered from the moment of BIG BANG.

The proposed model has the following basic assumptions:

a_0) the energy repartition in universe at the global scale this repartition is uniform enough to consider a constant energy density equal to its average value at a time t_u;

a_1) the analogue of a hydro-dynamical source is any amount of energy (larger then the energy of a graviton) emitting gravitons due to: 1) a higher energy level and 2) the expansion of universe;

On the Wind and Snow Loads Distribution and Their Effects on a K-Span Steel Arch Structure

Valentin BUTOESCU*[,1], Stefan NEBANCEA[1], Mihaela PETRE[2], Viorel ANGHEL[3],
Ioan STOICA[4]

*Corresponding author
[1]INCAS - National Institute for Aerospace Research "Elie Carafoli",
B-dul Iuliu Maniu 220, Bucharest 061126, Romania
butoescu.valentin@incas.ro*, nebancea.stefan@incas.ro
[2]AEROSPACE CONSULTING, B-dul Iuliu Maniu 220, Bucharest 061126, Romania
petre.mihaela@incas.ro
[3]"POLITEHNICA" University of Bucharest, Strength of Materials Department,
Splaiul Independentei 313, Bucharest 060042, Romania
vanghel10@gmail.com
[4]dipl. Engineer, ioan stoica@yahoo.com

Key Words: *Wind and Snow Loads, Vortex Method, Vaulted Roofs, K-Span Arch Structure*

This paper deals with the study of several load cases concerning the wind loads, snow loads and combined wind and snow loads distribution acting on a K-span arch which is used for buildings and roofing structures (fig. 1).

These kinds of structures are used because they are easy to achieve, fig. 2. Structural designer needs load distributions or design aerodynamic coefficients in order to assess the strength, stress and stability behaviour of such a structure.

The influence of the snow presence on the wind load distribution on a cylindrical arched roof is analysed using a vortex method. The shape of the snow drifts that were used for the aerodynamic calculations is presented in fig 3.

Fig.1 Roofing structures K-span arch

The flow separation and wake development are simulated by a vortex dynamics scheme.

Then using a simple beam model of the analysed structure the effects of such type of loads are assessed in terms of maximum bending moment, horizontal thrust, vertical loads and positions of the critical sections.

Determination of the potential spectrum in a wind tunnel nozzle by means of singularities methods

Alexandru DUMITRACHE*[,1] and Tudor Corneliu IONESCU[1]

*Corresponding author
[1]"Gheorghe Mihoc-Caius Iacob" Institute for Mathematical Statistics and Applied Mathematics of the Romanian Academy
Calea 13 Septembrie 13, 050711 Bucharest, Romania
alex_dumitrache@yahoo.com*, td.c.ionescu@gmail.com

Abstract: *In this paper, analytic and numerical results of the study of a potential flow through a nozzle in a wind tunnel are presented. The flow, considered axial-symmetric, is modelled by a distribution of vortex-ring type singularities. Finally, analytic distributions of the nozzle velocity are obtained together with the streamlines spectrum for several geometric forms of nozzle.*

1. INTRODUCTION

In this paper, the mathematical formulation and the numerical solution for determining the flow spectrum through a nozzle in an aerodynamic tunnel is presented. The velocity field of a flow can be computed using the potential model of an axial-symmetric flow. The differential equation of the stream function is given. A streamline of this function approximates the contour of the nozzle through the least squares method. The boundary values problem of an elliptical partial differential equation is the numerical problem that needs to be solved. A PC software has been developed and tested for several forms of nozzle, for which the streamlines have been computed.

2. PHYSICAL ASSUMPTIONS AND THE STREAM FUNCTION EQUATION

The nozzle is assumed axial-symmetric. Hence, the flow itself is axial-symmetric. In a wind tunnel, the vertical variation of the density is neglected.

The fluid is assumed incompressible and inviscid. Since the flow is potential, each component of the vortex vector $\Omega = \operatorname{rot}\vec{c}$ must be zero inside the domain of physical validity of the flow. In this paper, the vector field of the velocity $\vec{c}$, in a cylindrical coordinate system, has the following configuration: $\vec{c} = \{c_r(r,z);\ 0;\ c_z(r,z)\}_{\langle e_r, e_\varphi, e_z \rangle}$, since the tangential component, c_φ, is zero and the remaining two components do not depend on φ.

The singularities (sources or vortices) are not in the physical validity domain of the flow but are outside of this domain.

The continuity equation of this condition is: $\operatorname{div}\vec{c} = 0$. This equation in cylindrical coordinates is satisfied by the stream function Ψ, when $c_r = +(1/r)(\partial\Psi/\partial z)$ and $c_z = -(1/r)(\partial\Psi/\partial r)$ [1].

In cylindrical coordinates, the rotor components of the velocity vector are:

$$\operatorname{rot}\vec{c} = \left\{0;\ \frac{1}{r}\frac{\partial^2\psi}{\partial z^2} - \frac{\partial}{\partial r}\left(-\frac{1}{r}\frac{\partial\Psi}{\partial r}\right);\ 0\right\} \tag{1}$$

By the assumption that the flow is potential, each component of the vortex vector must be zero. The r and z components are satisfied identically. From the condition for component ϕ, the differential equation for the stream function is obtained

$$\frac{\partial^2\psi}{\partial r^2} - \frac{1}{r}\frac{\partial\psi}{\partial r} + \frac{\partial^2\Psi}{\partial z} = 0 \tag{2}$$

This is a linear, homogeneous, second order PDE. Since the equation is linear, the superposition principle can be used to solve it.

Solutions of the stream function. A function that satisfies the differential equation (2) is the stream function for parallel flows. For a flow with the velocity c_∞, in the positive direction of the z axis, the stream function

Turbulent Boundary Layer at Large Re

Horia DUMITRESCU[1], Vladimir CARDOS*[,1], Alexandru DUMITRACHE[1], Ion MALAEL[2]

*Corresponding author
[1]"Gheorghe Mihoc – Caius Iacob" Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy,
Calea 13 Septembrie no. 13, 050711 Bucharest, Romania
dumitrescu.horia@yahoo.com, v_cardos@yahoo.ca*, alex_dumitrache@yahoo.com
[2]National Research and Development Institute for Gas Turbine, COMOTI, Bucharest, Romania,
220D, Iuliu Maniu Avenue Bucharest 061126, Romania
ion.malael@comoti.ro

Abstract: *The startup of any flow is a kind of the impact process where there is momentum exchange between two colliding bodies, i.e. the flow and its boundary surfaces. Within a short time of contact in the post-impact shear flow, the two main effects are triggered by the flow-collision: dramatic redistribution of the momentum and vorticity followed by the shear stress/viscosity change in the micro-structure of the fluid. The disturbance of the flow at the impact induces the concentration of the bounded flow in the form of point-vortices or vorticity balls where waves are emitted and propagate through flow field. The key mechanism of propagation of these internal vorticity waves is the result of self-induced motion of concentrated boundary vorticity.*

Key Words: *Boundary layer, Velocity waves, Transitional flows.*

1. INTRODUCTION

The historic way of Fluid Dynamics is crossed through two major crises: the D'Alembert paradox-small D crisis [1], at the middle of the XVIII century, and the present big T–turbulence crisis [2].

As the D paradox was solved by the change from the concept of ideal fluid to the concept of the Newtonian (linear) viscous fluid, it is naturally to say that the T paradox/ crisis is a paradigmatic nature one, resulting from the use of the Newtonian fluid, a concept too rigid (linear and constant viscosity) to describe intricate turbulent motions.

The main drawbacks of the Newtonian fluid are two: firstly, the impossibility to describe the concentrated boundary vorticity, i.e. its creation, and secondly, the impossibility to reflect the change from any one state of microstructure of fluid to another for different states of flow.

Mechanically, the Newtonian fluid can describe only some simple flows (sparse vorticity and constant viscosity), analogous to rigid body motion with both constant acceleration and mass, in contrast to the turbulent flow involving variations of both concentration of boundary vorticity and viscosity.

The difference between the concept of Newtonian viscous and the thixotropic/non-linear viscous fluid is illustrated in the case of a start up flow due to an impulsively moved plane boundary.

The conventional approach is based on the mechanism of vorticity creation at wall by a diffusion process (Prandtl-Lighthill), [3]

$$\frac{\partial u}{\partial t}=\nu\frac{\partial^2 u}{\partial y^2},\quad t=0:\omega=-\frac{\partial u}{\partial y}=0\,,$$

together with the initial condition

$$u\left(y,0\right)=0,\,\omega\left(0,0\right)=-\frac{\partial u}{\partial y}=0,\,y>0$$

and the boundary conditions

$$u\left(0,t\right)=U,\,u\left(\infty,t\right)=0,\,t>0$$

The concept of Newtonian/linear viscous fluid assumes:

Index

SCIENTIFIC COMMITTEE of the Conference

- **Dr. Ruxandra BOTEZ**, École de technologie supérieure, Université de Quebec, Montreal, Canada
- **Dr. Gabriela MARINOSCHI**, ISMMA – Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy “Gheorghe Mihoc - Caius Iacob”, Bucharest, Romania
- **Dr. Adriana NASTASE**, Aerodynamik des Fluges, RWTH - Aachen, Germany
- **Dr. Ruxandra STAVRE**, Institute of Mathematics “Simion Stoilow” of the Romanian Academy, Bucharest, Romania
- **Dr. Sanda CLEJA-TIGOIU**, University of Bucharest, Romania
- **Dr. Mihai ARGHIR**, Université de Poitiers, France
- **Dr. Florin DIMOFTE**, Tribology Laboratory The University of Toledo, Office at NASA Glenn Research Center, Ohio, USA
- **Dr. Stefan BALINT**, West University of Timisoara, Romania
- **Dr. Corneliu BERBENTE**, “POLITEHNICA” University of Bucharest, Romania
- **Dr. Sterian DANAILA**, “POLITEHNICA” University of Bucharest, Romania
- **Dr. Horia DUMITRESCU**, ISMMA – Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy “Gheorghe Mihoc - Caius Iacob”, Bucharest, Romania
- **Dr. Horia ENE**, Institute of Mathematics “Simion Stoilow” of the Romanian Academy, Bucharest, Romania
- **Dr. Constantin FETECAU**, “Gheorghe Asachi” Technical University, Iasi, Romania
- **Dr. Dorel HOMENTCOVSCHI**, Binghamton University, USA
- **Acad. Marius IOSIFESCU**, ISMMA – Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy “Gheorghe Mihoc - Caius Iacob”, Bucharest, Romania
- **Dr. Mircea LUPU**, University “Transilvania”, Brasov, Romania
- **Dr. Dan MATEESCU**, McGill University, Montreal, Canada
- **Dr. Alexandru MOREGA**, “POLITEHNICA” University of Bucharest, Romania
- **Dr. Catalin NAE**, INCAS – National Institute for Aerospace Research “Elie Carafoli”, Bucharest, Romania
- **Dr. Cornel OPRISIU**, INCAS – National Institute for Aerospace Research “Elie Carafoli”, Bucharest, Romania
- **Dr. Titus PETRILA**, “Babes-Bolyai” University, Cluj-Napoca, Romania
- **Dr. Dan POLISEVSCHI**, Institute of Mathematics “Simion Stoilow” of the Romanian Academy, Bucharest, Romania
- **Dr. Constantin POPA**, Ovidius University of Constanta, Romania
- **Dr. Sorin RADNEF**, INCAS – National Institute for Aerospace Research “Elie Carafoli”, Bucharest, Romania
- **Dr. George SAVU**, COMOTI – National Research and Development Institute for Gas Turbines, Bucharest, Romania
- **Dr. Ion STROE**, “POLITEHNICA” University of Bucharest, Romania
- **Dr. Victor TIGOIU**, University of Bucharest, Romania
- **Dr. Ioan URSU**, INCAS – National Institute for Aerospace Research “Elie Carafoli”, Bucharest, Romania

ORGANIZING COMMITTEE

Prof. Dr. Corneliu BALAN

U.P.B. – "Politehnica" University of Bucharest

E-mail: balan@hydrop.pub.ro

http://www.pub.ro

Prof. Dr. Adrian CARABINEANU

University of Bucharest

E-mail: acara@fmi.unibuc.ro

http://www.unibuc.ro/

Dr. Stelian ION

I.S.M.M.A. – Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy "Gheorghe Mihoc - Caius Iacob"

E-mail: ro_diff@yahoo.com

http://www.ima.ro/

Prog. Elena NEBANCEA

INCAS – National Institute for Aerospace Research "Elie Carafoli"

(under the Aegis of the Romanian Academy)

E-mail: nebancea.elena@incas.ro

http://www.incas.ro

Contents

ISSN 2067-4414
ISSN-L 2067-4414

ISSN National Center
Romanian National Library

BUCHAREST
2016