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

Proceedings
of
the XXXIInd “Caius Iacob” National Conference
on
Fluid Mechanics and its Technical Applications

2009, October 16 – 17

Bucharest, Romania

Organizers

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

University of Bucharest

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

“Politehnica” University of Bucharest

BUCHAREST

2009

Proceedings

of

the XXXIInd "Caius Iacob" National Conference on FLUID MECHANICS and its TECHNICAL APPLICATIONS

Bucharest, Romania,
October 16 – 17, 2009

Sessions:

Mathematical Modeling & Numerical Simulations
Dynamical Systems
Technical Applications

Editor-in-chief:

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

Registration code: **ISSN 2067 – 4414**

ISSN National Center

ROMANIAN NATIONAL LIBRARY

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

Proceedings
of
the XXXIInd “Caius Iacob” National Conference
on
Fluid Mechanics and its Technical Applications

2009, October 16 – 17

Bucharest, Romania

Organizers

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

University of Bucharest

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

“Politehnica” University of Bucharest

BUCHAREST

2009

Foreword
In memoriam Lazăr Dragoş

On April 2, 2009 Professor Lazăr Dragoş, passed away. Even if during his last years his presence in the public life was more and more discreet, because of his illness, his death was a great loss to the community of the mechanicians and mathematicians. Professor Dragoş' last years of active life were a real race against the clock. Although increasingly weaker, the teacher had accomplished the mission: of gathering his latest aerodynamics research in a monograph. His monumental monograph *Metode matematice in aerodinamica* first was printed by the Academy Publishing House and then by Kluwer Publishing, in English, under the title *Mathematical Methods in Aerodynamics.* It is a synthesis of original scientific results obtained by Professor Lazăr Dragoş during the last quarter-century of his life. The original fundamental solutions obtained for various forms of linear aerodynamics equations , the original analytical and numerical solutions of integral equations utilized in the study of aerodynamics problems give the monograph a special status compared to other specialized studies The professor's greatest wish was that this book be a tool, a legacy left to the Romanian school of fluid mechanics. The Monograph was the result of a long educational activity during which the teacher shared his ideas, accepting in turn the ideas of his students and collaborators We give in this connection an extract from the preface to the English language edition: "*In every work one finds, in a certain measure, both the achievements of the predecessors and of the researchers contemporaneous with the author. Among the people which have directly collaborated with me, I have to mention first my professors Victor Vâlcovici and Caius Iacob, who introduced me in the field of aerodynamics. I also mention my younger colleagues Nicolae Marcov, Liviu Dinu, Dorel Homentcovschi, Adrian Carabineanu, Victor Ţigoiu, Vladimir Cardoş, Gabriela Marinoschi, Stelian Ion and Adrian Dinu. They were my students at the University of Bucharest, but I learned a lot from their papers. Some of them were my fellow-workers in the aerodynamics research, many of them stimulated me with their youth and their way of thinking in our seminars from the Faculty of Mathematics of the University of Bucharest. I am very grateful to all of them".*

But before reaching this climax it was a long way. Let us recall several stages of this way. The young Lazăr Dragoş, a last year student at the high school won the second prize at the National Mathematics Olympiad (the first prize was won by Ciprian Foiaş). He became then a student of the Faculty of Mathematics and Physics from the University of Bucharest. On the notice board of the Department of Mechanical Engineering he read the Syllabus of theoretical mechanics course held by Professor Victor Vâlcovici and decided to dedicate him to the study of mechanics.

This moment will mark his entire activity. After finishing the college, Lazăr Dragoş passed through all the stages of the educational activity at the Chair of Mechanics of the University of Bucharest. He achieved his first important scientific results in the field of the magneto fluid dynamics and the synthesis of these results is given in monographs *Magnetodinamica fluidelor,* published by the Academy Publishing House, in 1969 and *Magneto Fluid Dynamics*, published by Abacus Press in 1975. Moreover his doctoral thesis topic was the theory of thin wing in magneto-aerodynamics. The area of interests of the young researcher and teacher Lazăr Dragoş then become increasingly larger. He taught courses and led seminars in general mechanics (Newtonian mechanics, analytical mechanics and relativistic mechanics), continuous media mechanics (hydrodynamics and elasticity theory), fluid mechanics, magnetofluidynamics, aerodynamics. He also held lectures on the theory of free surface-fluid motion, dynamics of rivers, oceans and atmosphere, kinetic theory of gases and plasma, etc.. He published fundamental works laudatory cited in specialized international literature of the following areas: general mechanics, fluid mechanics, elasticity theory, magneto-fluid dynamics, magneto elasticity and magneto-thermoermoelasticity, aerodynamics.

The highest points of these scientific and didactic concerns were his two monographs, namely *Principles of analytical mechanics, Technical Publishing House, 1976 and Principles of continuous media Mechanics, Technical Publishing House 1986.*

Also interesting to study is the way Professor Lazăr Dragoş worked with the others. If the young researcher Lazăr Dragoş, proud and aware of his value declined any guidance and develops a thesis without having a scientific leader, later, the experimented researcher Lazăr Dragoş, has discovered the valences of the teamwork. Professor Dragoş collaborated among others with N. Marcov in the study of electro conductive viscous fluids, with D. Homentcovschi in magneto-hydrodynamics problems, with D. Homentcovschi, A. Dinu and A. Carabineanu in aerodynamics problems. He guided the work of several researchers like V. Ţigoiu (kinetic theory of gases), G. Marinoschi (theory of magneto-hydrodynamic generators), S. Ion (theory of motion with shock waves), etc... After the death of the Academician Caius Jacob, Professor Lazăr Dragoş, himself a member of the Romanian Academy from 1991, becomes the spiritual leader (probably it is the most appropriate term) of mathematicians interested in fluid mechanics. He aims to gather in a large monograph the main achievements of the Romanian specialists in Fluid Mechanics having university studies. For this purpose he published in 1998 the first Volume of his monograph Fluid Mechanics and invited a number of researchers and teachers to collaborate on writing the other volumes. But time had no patience. The Professor' s health condition started to worsen and his energy no longer allowed him to continue this project, so he focused on developing the monograph already mentioned *Mathematical methods in aerodynamics.*

Looking back in time we can not dissociate the personality of the Academician Lazăr Dragoş from these of his teachers, the Academicians Victor Vâlcovici and Caius Iacob whom he was first a student and then colleague and collaborator. If the name of professor Vâlcovici is linked to the international recognition of the Romanian science during the interwar period and the name of Professor Iacob is related to the intellectual resistance against the dictatorship by means of science, the name of Professor Dragoş is linked to devotion to science in a society where diplomas are not naturally obtain through work and researchers are dismissed from their jobs. Perhaps the destiny has made a gift to Professor Dragoş, carrying him away and exempting him to be a witness of the degradation of the scientist current condition. Now, all those whom the teacher considered (rightly or not) as followers of his work have a duty to keep the values of honest work in teaching and research activities and not leave the Romanian School of Mathematics-Mechanics to become a mere memory.

Professor dr. Adrian Carabineanu, Faculty of Mathematics and Computer Science, University of Bucharest.

CONTENTS

- Maria Alexandrescu, Nicuşor Alexandrescu – Ice accretion simulation on NACA-0012 ……….. 1
- Irina Carmen Andrei, Mihai Barbelian, Gabriela Liliana Stroe – Jet engines thrust control system analysis ……………………………………………………...………………………. 16
- Daniela Baran, Nicolae Apostolescu – Numerical aspects in dynamic response of an elastic plane ………………………………………………………………………………………. 29
- Corneliu Berbente, Sorin Berbente - An extension of Hugoniot – Rankine relations for reacting and nonreacting flows considering the variation of specific heats with temperature …………….. 38
- Andreea Bobonea – Computational fluid dynamics analysis of flow over a re-entry blunt body ... 46
- Valentin Butoescu – An elastic model of a textile surface …………………………………...... 55
- Mircea Dimitrie Cazacu, Dan Andrei Andreian, Bogdan Andreian – Aquaplaning of airplane tyre ...…………………………………………………………………………………………… 63
- Cristian Cârlănescu, Ionuț Porumbel, Constantin Eusebiu Hrițcu – Linear eddy model numerical simulation of the turbulent flow in a round turbulent jet …………………………………………. 73
- Aurel I. Chirilă, Alexandru M. Morega – Electric motor frame cooling; a finite element analysis 83
- Stefan-Gicu Cruceanu – Arclength continuation for implicitly defined curves ...………………... 90
- Alexandru Dumitrache, Florin Frunzulică, Horia Dumitrescu – A Numerical analysis of combustion process in an axisymmetric chamber ……………………………………………... 102
- Florin Frunzulică, Alexandru Dumitrache, Horia Dumitrescu, Vladimir Cardoş – Unsteady free-wake vortex particle model for horizontal axis wind turbines ……………………………………. 109
- Stelian Grădinaru – Self-propulsion of thin profiles; an analytic approach ……………………… 115
- Ionel Cristinel Iorga – The effect of saturation on the longitudinal command in the approaching phase of flight for landing …………………………………………………………………… 125
- Gheorghe Juncu, Aurelian Nicola and Constantin Popa – A numerical study of laminar flow past two circulating spheres in tandem………………………………………………………………… 132
- Ion Lăncrănjan, Ionuț Brînză – Applications of visualization methods in aeronautics and civil engineering ……………………………………………………………………………………... 138
- Alexandru M. Morega, Alin Dobre, Mihaela Morega, Daniel Mocanu – Finite element modeling of blood flow in an arterial bifurcation ………………………………………………………… 148
- Cătălin Nae – Noise metric for high-lift-low-noise concept validation …………………………... 152
- Mihai Leonida Niculescu, Mircea Dan Ionescu, Marina Andrei, Cristian Cercel – Analysis of aerodynamic field and noise from a ducted wind turbine ………………………………………… 159

CONTENTS

- Marin V. Petre, Alexandru M. Morega – A numerical study of the heat transfer by the aluminum electrolysis cell shell …………………………………………………………………………… 172
- Dumitru Popescu, Alin Gabriel Popescu, Ecaterina Mărieş – Mathematical modelling of a giant unilamellar vesicle under positive osmotic pressure ……………………………………………... 179
- Octavian Preotu, Alexandru Dumitrache, Horia Dumitrescu – Similar solution for a Coanda flow and applications to directed synthetic jets …………………………………………………... 193
- Virgil Stanciu, Irina Carmen Andrei – Optimization and stability issues in simulating the running regimes of the jet engines ………………………………………………………………... 206
- Claudia Alice State – The influence of saturation relative to the aileron control in the roll motion 216
- Richard Şelescu – Intrinsic analytic study on an isoenergetic flow of a compressible fluid Part 1: “Isentropic” and “zero work” surfaces in aero-gas dynamics……...................................... 224
- Richard Şelescu – Intrinsic analytic study on an isoenergetic flow of a compressible fluid Part 2: Extension to some cases in magneto-plasma dynamics …………………………………... 239
- Claudia Timofte – Asymptotic analysis of a degenerate evolution system ………………………. 259
- Index ……………………………………………………………………………………………... 265

Proceedings of the XXXIInd "Caius Iacob" National Conference on Fluid Mechanics and its Technical Applications
2009, October 16 – 17, Bucharest, Romania
Maria Alexandrescu, Nicuşor Alexandrescu
Ice accretion simulation on NACA-0012

Ice accretion simulation on NACA-0012

Dr. Mat. Maria Alexandrescu*, Fiz. Nicuşor Alexandrescu*
"Elie Carafoli" National Institute For Aerospace Research – INCAS, 061126, Bucharest, Romania

Abstract: In the article, the problem, of in-flight ice accumulation on airfoil with NACA-0012 section, is presented. The our analysis is a developement of the IBL (interactive boundary-layer) technique of Cebeci [2]. The IBL technique uses a inviscid/viscous boundary-layer iteration scheme where the boundary layer is calculated "under" the inviscid solution, and then the boundary-layer results are used to update the wall boundary conditions and a new inviscid solution is calculated. The calculation of flow field use the Hess-Smith panel method [1]. The calculation of droplet trajectories is effectued by Lagrange method.
The convective heat transfer coefficient distributions around the airfoil are calculated using Boundary-Layer Method. The final step consists of establishing the thermodynamic balance and computing ice accretion rates. Ice shapes are computed for realistic flight conditions. Predicted ice shapes are in good agreement with experimental shapes reported in the literature.

1. Water droplet trajectory calculations

(*a*) *Problem description and assumptions*

The first stage in any icing analysis is to determine where and at what rate the cloud water droplets are deposited on the surface of the body (e.g. wing or fuselage) under investigation, [2]. There are two main types of trajectory calculation in use today.

The first and most frequently adopted method is a traditional Lagrangian formulation in which the trajectory of an individual water droplet is tracked from a specified starting point upstream of the body (usually five to seven chord lengths ahead of the body).

The second and more recently adopted approach is Eulerian, in which the volume fraction is computed at the same node positions at which the aerodynamic parameters are known. No individual particles are therefore tracked in this approach and the answer leads directly to a measure of the catch effciency.

For both methods, either a flow solution or a means of calculating the aerodynamic flow is necessary. Methods adopted depend on the application and range from simple panel methods through to full Navier-Stokes solvers.

(*b*) *Governing equations*

The derivation of the equation for the droplet acceleration in the x-direction is shown below [3]. The derivation for the acceleration in the y- and z-directions for three dimensional calculations is very similar and therefore not shown. The primary assumptions on which the equations are based are as follows.
(1) The droplets are spherical and do not deform.
(2) There is no collision or coalescence of droplets.
(3) Turbulence effects may be neglected.
(4) The only forces acting on the droplet are due to aerodynamic drag, gravity and buoyancy.
(5) The water droplet concentration is sufficiently small for the droplets to have a negligible effect on the aerodynamic flow and therefore the air flow and water droplets may be treated as independent systems.

An illustration of the coordinate system and terminology employed in this note for the aerofoil and droplet is provided in figures 1 and 2.In the case of a single particle, where V_a is the local air velocity, u_a is the local air velocity component in the x-direction, v_a is the local air velocity component in the y-direction, V_d is the droplet velocity, u_d is the droplet velocity component in the x-direction, v_d is the droplet velocity component in the y-direction, u_{rel} is the relative air/droplet velocity in the x-direction, v_{rel} is the relative air/droplet velocity, V_{res} is the resultant of the relative velocities u_{rel} and v_{rel}.

* Tel.: +40 021 434 0083; fax: +40 021 434 0082
E-mail address: malex@incas.ro ; *URL:* http://www.incas.ro

Jet engines thrust control system analysis

Irina Carmen Andrei*, Mihai Barbelian**, Gabriela Liliana Stroe***

"POLITEHNICA" University of Bucharest, Faculty of Aerospace Engineering, "Elie Carafoli" Department of Aerospace Sciences; "Nicolae Tipei" Department of Engineering and Management in Aeronautics; Str. Gh. Polizu, nr. 1-7, sector 1, Bucureşti, 011061, ROMANIA.
**PhD, MSc AE, Lecturer*, **PhD Student, MSc AE, Lecturer, ***PhD Student, MSc AE, Teachnig Assistant;*
ICAndrei28178@gmail.com, irina.ac@aero.pub.ro

Abstract: The intent of this paper is to present the analysis of a jet engine's thrust control system. The running regimes of a jet engine, as well as the flight operational conditions are varying over large intervals ranging the flight envelope. The task fulfilled by the thrust control system is to provide the necessary amount of thrust required by the flight operational conditions, following the proper adjustment of the combustion fuel flow rate and the exhaust nozzle area. The design of the thrust control system can be done in two ways, such that the regulation process can either: (1) – maintain the thrust to a prescribed value, irregardless the external perturbations, or (2) – bring any deviation in engine thrust, back to zero as soon as possible, without crossing the compressor's surge margins or without exceeding the maximum allowable value of the turbine inlet temperature. The control strategy can be achieved by employing an optimal control law (1) or (2) of linear state feedback form, which can be obtained from the solution of a linear quadratic problem. The test case is a twin spool mixed flows turbofan, with afterburner, i.e. the F-100 turbofan. The dynamic response of the engine throughout the flight envelope can be properly adjusted by using an optimal control law.
2000 Mathematics Subject Classification: 37xx; 37Mxx; 37Nxx; 37N35; 37N30; 65xx; 65Pxx.
Keywords: jet engine, thrust control system, flight envelope/ operational conditions, control strategy, optimal control, dynamic response of the engine, jet engines, optimization, simulation of running regimes, analytical model, state and output equations, aerospace sciences.

1. Introduction

The study conducted within this paper aims to optimize the control of a jet engine, expressed in terms of jet engine performances (namely the thrust and the specific fuel consumption) versus the control vector; the influence of the flight conditions it is expressed by the appropriate matrices A, B, C and D of the mathematical model. There have been studied various control laws, such that numerical simulations have been done for the command given as: 1)- step function, 2)- ramp, 3) a combination of limited ramp followed by a step function, and 4)- impulse.

A development of the mathematical model for more complex construction has been pointed out.

The basic construction is represented by the single spool turbojet TR-1; the twin spool turbojet TR-2 is a construction featuring higher ability to adjust to the changes (i.e. perturbations) of flight operating conditions FC's. Then, the turbofan family (which includes: a)- high by-pass turbofans, such the latest GenX, Ge90, Trent 1000 and all category of Un-ducted Flow UDF engines, and b)- low by-pass turbofans, as the military F100/ F110/ EJ200) show the capability of better adjustment to the FC's, with lower values of specific fuel consumption with respect to the turbojet family. The turbofan constructions are either twin or triple spool, which means that the mathematical model must include an extra element to the input vector; usually it is the speed [rpm] of the second spool that drives the high pressure compressor HPC into rotation from the high pressure turbine HPT. For the case of the military engines, both turbojet TR-1/ TR-2 or low by-pass turbofan, the presence of an afterburner requires the addition of another element to the input vector, namely the afterburner fuel flow.

The test case is represented by the PW F100 turbofan (which is a low by-pass turbofan, with afterburner).

The simulation of the running regimes of the engine was represented by a linearized model of the jet engine, describes by the output and state equations.

Tel: +40 21 4023967, +40 723 859352, Fax: +40 21 3181007,
E-mail address: ICAndrei28178@gmail.com , *URL:* http://www.upb.ro, http://www.aero.pub.ro

Proceedings of the XXXIInd "Caius Iacob" National Conference on Fluid Mechanics and its Technical Applications
2009, October 16 – 17, Bucharest, Romania
Daniela Baran, Nicolae Apostolescu
Numerical aspects in dynamic response of an elastic plane

Numerical aspects in dynamic response of an elastic plane

Daniela Baran*, Nicolae Apostolescu**
*) **)*National Institute for Aerospaciace Research "Elie Carafoli", 220 Iuliu Maniu, Bucharest, Romania*

Abstract: In order to analyze the dynamic response of a flexible aircraft and landing gear we develop a family of simplified models: the first one is a two degrees of freedom model, the second is a rigid four degrees of freedom model and the third is a model with seven lumped masses based on the second one. For these models we built the equations of motion and solve them with a Runge-Kutta algorithm. Shock absorbers are usually characterized by non linear type of damping (for example quadratic) and shock absorbers use different types of damping linear and non linear. In this paper we combine different types of linear and non linear damping and stiffness for the considered family of simplified models. The main aim of this paper is to investigate stability aspects of the ordinary differential equations considered.
2000 Mathematics Subject Classification: 34 Ordinary differential equations, 37 Dynamical systems, 70 Mechanics of particles and systems.
Keywords: dynamic response, ordinary differential equations, stability of dynamic systems.

1. Introduction

Computational models have become essential to a successful design of modern and complex engineered systems such as an aircraft. The use of a computational model as a design tool allows searching a much larger parameter space and thus has lead to products with highly optimized performance and efficiency.

In order to analyze the dynamic response of an elastic aircraft and landing gear we develop simplified models: a model with two degrees of freedom, a model with four degrees of freedom and a model with seven degrees of freedom.

For all these models we built the equations of motion and solve them with a forth order Runge-Kutta algorithm. Both models are subjected to "round step" excitation [2], a time dependent function used to model the shock during landing.

Such a time dependent function to represent the landing impact is expressed by the following relation [2]:

$$f(\tau) = \gamma\left[1 - e^{-\gamma\tau}(1 + \gamma\tau)\right], \tag{1}$$

where γ is a parameter which describes the shock intensity (figure 1). In our stability analysis it represents an important parameter. In expresion (1) as it is considered in the reference [2], τ in a non dimensional time. In each case of the analysis we shall make the appropriate change of variable.

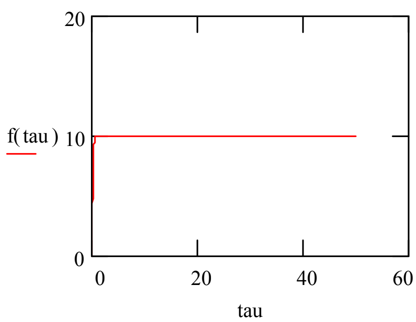

Figure 1. Round step excitation.

* Tel.: +40 021 434 0083; fax: +40 021 434 0082
E-mail address: dbaran@incas.ro ; *URL:* http://www.incas.ro

An extension of Hugoniot – Rankine relations for reacting and nonreacting flows considering the variation of specific heats with temperature

Corneliu Berbente*, Sorin Berbente**
"POLITEHNICA" UNIVERSITY OF BUCHAREST FACULTY OF AEROSPACE ENGINEERING
**) "Elie Carafoli" Department of Aerospace Sciences ; **) "Nicolae Tipei"t of Engineering and Management in Aeronautics*
Str. Gh. Polizu, nr. 1-7, sector 1, Bucureşti, 011061, ROMANIA.
* Prof. PhD, ** PhD, MSc AE, Lecturer * berbente@yahoo.com, * sun_so@yahoo.com

Abstract: The Hugoniot-Rankine relations for the ideal gases were obtained in case of constant specific heats, leading to hyperbolas in pressure-volume coordinates.Because the variation of the specific heats with temperature are rather complicated (for example, seven terms on three intervals, in case of NASA data), one uses., in general, numerical methods of calculations. In the following one presents a possibility to extend the Hugoniot-Rankine and other relations as well, by analitycal working up, preserving the form and representation, in case of variable specific heats. In this .way, a single equation in one variable has to be solved. One introduces dimensionless expressions for temperature and thermodynamic functions.
2000 Mathematics Subject Classification:"76 J 20" " 80 A 32"
Keywords: mean caloric capacity, dimensionless temperature, equivalent Mach number;

1. Introduction

One considers the 1 D flow of an ideal gas. In general, one deals with a gas mixture of components that cam be reacting or nonreacting. In cas of a reacting gas, the oxidant (ex. Air) and the fuel (ex. Hydrocarbon) are premixed, and a combustion wave occurs. For nonreacting flow a shock wave is produced at supersonic speeds, when from some reason the flow is slowed down.

By adopting the indices „1" and „2" for the initial and the final states respectively, the governing equations of motion are considering adiabatic flow:

$$\rho_1 u_1 = \rho_2 u_2 \equiv \dot{m}\,(mass\,conservation) \quad (1-a)$$

$$p_1 + \rho_1 u_1^2 = p_2 + \rho_2 u_2\,(momentum) \quad (1-b)$$

$$h_1(T_1) + \frac{1}{2}u_1^2 = h_2(T_2) + \frac{1}{2}u_2^2\,(energy) \quad (1-c)$$

In the above eqs. One denotes by ρ, p and T the density, pressure and temperature (Kelvin degree), u being the velocity and $\dot{m}$ the mass flow rate.

As ? the enthalpy *h (T)* of an ideal gas, one has:

$$h\,(T) = h\,(T^0) + \int_{T^0}^{T} c_p\,(T)\,dt, \quad (2)$$

$c_p\,(T)$ being the specific heat at constant pressure and T^0 a reference temperature. For convenience, one takes,

$$T^0 = T_1. \quad (3)$$

2. The problem solution for c_p = const.

For constant specific heat c_p = const. The eqs. (1 – c) and (2) are reduced to to the simplified equation:

$$c_p T_1 + \frac{1}{2}u_2^2 + q = c_p T_2 + \frac{1}{2}u_2^2 \quad (c_p = const.) \quad (4)$$

$$q = h_1(T_1) - h_2(T_1); \quad (5)$$

** Tel: +40 21 4023967, Fax: +40 21 3181007,
E-mail address: berbente@yahoo.com, *URL:* http://www.upb.ro, http://www.aero.pub.ro

Computational fluid dynamics analysis of flow over a re-entry blunt body

Andreea Bobonea[1]
Institutul National de Cercetari Aerospatiale „Elie Carafoli``, Bd. Iuliu Maniu Nr. 220, sector 6, Bucuresti, Romania

Abstract: This work focuses on the study of the flow in the vicinity of blunt bodies during their re-entry into the Earth's atmosphere using computational fluid dynamics techniques. The study makes use of flow solver Fluent 6.2 to perform a series of unsteady state flows around 2D basic shapes (circle, ellipse and double ellipsoid). These studies demonstrate the importance in understanding the effects of shockwaves and illustrate how any minor alteration to the shape of the body has great implications over the flow.
2000 Mathematics Subject Classification: 76, Fluid mechanics
Keywords: blunt body, computational fluid dynamics, hypersonic, shockwaves, unsteady flow

1. Introduction

The development of modern space transportation systems requires an enormous effort in the field of aerothermodynamics. Firstly, from this discipline it will be expected to achieve information regarding local and global forces and moments which can be further used in the study of maneuverability and flight control of the spacecraft at considered ranges of altitudes and velocity. Secondly, knowledge about the thermal loads (heat flow, temperature distribution at the wall) governing the construction of a thermal protection system (TPS) plays an important role in the design of a hypersonic shuttle.

In addition to information obtained from the wind tunnel and free flight measurements, the predictions using numerical methods are powerful means for solving the above mentioned problems. If in aerodynamic tunnels for a hypersonic flow one cannot obtain valid information for a shuttle configuration with realistic dimensions that is because of specific parameters, Reynolds number, Mach number and side of movement which can not be simultaneously met. Aerodynamic tunnels are an important part of basic phenomena research and validation of predictive numerical methods.

2. Method

With the development of computational aerodynamic techniques (Euler-Navier-Stokes) and the increased computing, computational fluid mechanics numerical simulations allow the aircraft aerodynamic analysis taking into account all the phenomena occurring in such a flow with results that are very close to experimental data. Fluent commercial software is one of the fluid mechanics programs that have been successfully tested to simulate many types of flows and geometries, from incompressible flows (low speed) to compressible flows (transonic and supersonic regime).

To study the thermal effects arising from the spacecraft entering the atmosphere we considered 3 basic configurations for which the numerical simulations were carried out.

2.1 Geometrical modeling

The geometrical configurations (circle, ellipse, and double-ellipsoid) were created with the Gambit preprocessor. Control volume has a length of 15 m and the circle has a radius of 0.5 m. Ellipse has a radius greater than 0.25m and the radius of 0.5m. To refine the solution, the grid was adapted twice in the double ellipsoid case. The small ellipse has the radius of 0.1m and 0.2m and the big ellipse has the radius of 0.3m and 0.61m.

[1] * Telefon: (+40)-(21)-4 34.00.83; Fax: (+40)-(21)-434.00.82
E-mail address: abobonea@incas.ro ; *URL*: http://www.incas.ro

An elastic model of a textile surface

Valentin Butoescu*

Department of Aerodynamics "Elie Carafoli" National Institute for Aerospace Research – INCAS, 061126, Bucharest, Romania

Abstract: An elastic model of a textile surface is presented. This model is intended to be used for aeronautical or nautical applications (parachutes, paragliders, sails). The model is based on the concept of a "textile atom", i.e. a textile surface element is represented by a single particle that is linked to the other particles by elastic bonds with dampers. Two of the elastic bonds represent in fact the warp and weft. Moreover, each particle is linked to the others by bias bonds. This model is of course highly non-linear. Mathematically it is equivalent to a large system of differential equations with known initial conditions, representing the motion of the particles. In fact we seek for the equilibrium state, that is the particle positions for $t \to \infty$. The system is solved numerically using a simple Euler method. The results are realistic.
2000 Mathematics Subject Classification: 70
Keywords: elastic model; textile surface; differential equations; equilibrium state; Euler method.

1. Introduction

The first attempt to model a textile surface was done by Chaplygin. His model regards the inextensible textile materials and it corresponds to the inextensible string theory in 1D. However in our applications (parachutes, paragliders, sails) we are rather interested in elastic textiles. There are two ways a textile surface could be modeled: the elastic coupled particle model and the continuum model [1]. Examples of particle models could be found for instance in Puiseux's work [2], and also in a paper written by Villard and Borouchaki [3]. The continuum model is analyzed in [8] and then compared with the particle model. The present paper describes a particle model method, which has been elaborated under the Contract no. 115 with CNMP.

The author has chosen the particle model because it could be easily used in connection with some other methods encountered in Aerodynamics [2]. On the other hand the model has the advantage of being very close to a physical interpretation. One can argue that there is not necessary to solve the system of differential equation to find the equilibrium position of the textile surface. However, we rather seek for a solution that that is applicable to the dynamic problem of parachute deployment as well.

2. Textile material modeling

Consider a textile surface which is not acted by any forces. Its structure consists of a lattice of threads: the warp and weft. In this equilibrium state, the warp and the weft are perpendicular to each other. The treads of warp and weft make up equidistantly networks: there is constant step Δu_0 for the warp and a constant step Δv_0 for the weft. The textile surface has a certain superficial density, ρ_t which is constant if the cloth is not loaded. Let us consider a certain rectangular textile surface having the length $L_t=n\times\Delta u_0$ and the width $l_t=n\times\Delta v_0$, $n\in N$. So the network has n^2 rectangular eyes. Fig. 1 presents such a typical eye.

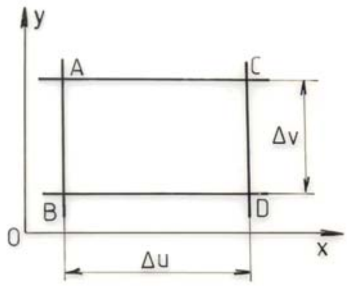

Fig. 1 An eye of textile material

*Tel.: +40 021 434 0083; fax: +40 021 434 0082
e-mail address: vbutoesc@incas.ro, valentinbutoescu@yahoo.com; *url:* http://www.incas.ro

Aquaplaning of airplane tyre

Mircea Dimitrie Cazacu*, Dan Andrei Andreian, Bogdan Andreian,
Polytechnic University, Bucharest 060042, Splaiul Independentei 313, cazacumircea@yahoo.com, andreian.dan@gmail.com

Abstract: Because this dangerous phenomenon have appeared for the first time in the case of the airplane landing on an aerodrome covered with a water pellicle, we studied the three-dimensional flow mathematical problem for the high pressure tires, using the relative liquid flow with respect to the mobile Cartesian trihedron, solidary with the airplane wheels, the liquid flow being permanent in this case and the boundary conditions being more simples to put of mathematical point of view.
One considers the equation system with partial differentials, corresponding to the relative flow and transformed for the dimensionless variables and functions, considering the three velocity components of the liquid steady flow and the high pressure variation deduced by the experimental researches tacked from the technical literature.
To solve these equations we establish the specific boundary conditions and we used the iterative numerical calculus, developing the three velocity component functions in finite Taylor's series for a three-dimension grid, with two constant steps and the third step variable with the tyre spot length, studying also the numerical solution stability. One presents some obtained results concerning the streamline function for the different surfaces in the liquid pellicle, marked by the computer program.
2000 Mathematics Subject Classification: "76 A 20"; "74 K 35"; "35 Q 75".
Keywords: "Aquaplaning"; "Tyre aquaplaning"; "Airplane landing".

Because this three-dimensional phenomenon (fig.1) have appeared for the first time in the case of airplane landing on a aerodrome covered with a water pellicle and the problem of mathematic study is more simple in the conditions of mobile trihedron, solidary with the airplane wheels [1], such of the boundary condition laying at the liquid domain limits, comprised between the tyre and the aerodrome covered with water, as also to the steady character of the relative motion [2][3], we shall consider the Navier-Stokes equation system with partial differentials, corresponding to the relative flow trihedron.

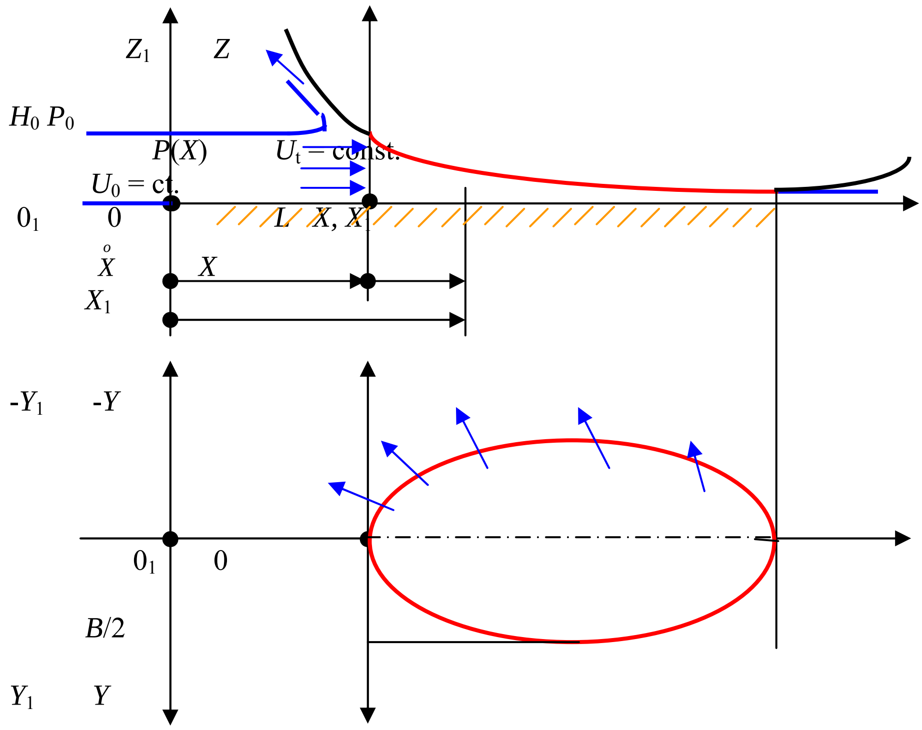

Fig. 1. Three-dimensional model of the airplane tyre hydroplaning flow

* *E-mail address:* cazacumircea@yahoo.com

Linear eddy model numerical simulation of the turbulent flow in a round turbulent jet

Cristian Cârlănescu**, Ionuț Porumbel**, Constantin Eusebiu Hritcu
National Research and Development Institute for Gas Turbines – COMOTI, 220D, Iuliu Maniu Ave. 061126, Bucharesti, 6, România
e-mail: ionut.porumbel@comoti.ro, web site: http://www.comoti.ro

Abstract: The paper presents an application of the Linear Eddy Model for the numerical simulation of a multi-species, hot, round turbulent jet formed at the exhaust of a gas turbine. The numerical algorithm employed herein is one-dimensional, on a radial domain that is convected axially at a prescribed velocity, determined from the round turbulent jet scaling laws. The radial velocity in the domain is determined from the continuity equation. The temperature and major chemical species diffusion equations are solved at the smallest scales of the turbulent problem, while the effect of turbulence is implemented through the Linear Eddy Model method, that interrupts the numerical integration at time intervals determined from inertial range scaling laws. The position where the turbulent eddies are occurring is chosen from an original probability distribution function designed to reproduce accurately the physics of the turbulent jet. The numerical results are validated against experimental measurements carried on in the exhaust jet of a Garrett 30 – 67 gas turbine engine.
Keywords: Gas turbine, turbulent jet, Linear Eddy Model, numerical simulation, experimental measurements.

1. Introduction

In principle, a turbulent jet is produced by a Newtonian fluid flowing steadily through a nozzle of diameter d_0, into a quiescent surrounding fluid, producing a flat-topped velocity profile at the nozzle, of mean velocity U_0. The flow is statistically stationary and axisymmetric, hence flow statistics depend on the axial (x) and radial (r) co-ordinates, but are independent of time (t) and of the circumferential co-ordinate (θ)[1]. Non-reactive turbulent jets have been studied for a long time, both theoretically and experimentally, and a multitude of earlier work can be quoted in this field.

Thus, Schlichting[2] solved for the first time the boundary layer momentum and continuity equation for the round turbulent jet in 1933. Pope[1] describes in detail the equations governing the turbulent jet. Hussein and others[3] and Wygnanski and Fiedler[4] presented experimental measurements of axial and radial profiles of mean velocity in a round turbulent jet, at various high Reynolds number and for different geometries. The spreading rate of the round turbulent jet and its velocity decay constant has been measured, with different methods, but with similar results, by Panchapakesan and Lumley[5,6], Hussein and others[3] and Mungal and Hollingsworth[7].

From a numerical problem, the turbulent flow in the exhaust plume of an aviation gas turbine engine has been studied by Wu and Menon[8,9], using the Linear-Eddy model (LEM) in a similar approach. The numerical algorithm employed for the numerical study herein has been applied previously by Porumbel and Menon[10] for the study of soot formation in turbulent jet diffusion flames. The LEM technique has been applied earlier in a number of numerical studies: by Menon and Jou[11], in a ramjet combustor, and by Calhoon and Menon[12],and Menon and others[13] in jet diffusion flames.

This paper follows the work of Wu and Menon[8,9] and proposes a simplified LEM solution algorithm for the multiple species round turbulent jet, able to predict both the axial and the radial distribution of chemical species mass fractions and temperature in the jet. The algorithm is applied on a one – dimensional radial domain that is convected downstream at a prescribed velocity. The molecular diffusion is exactly solved in the domain using the diffusion equation discretized at the appropriate scales, while the turbulence effect is implemented concurrently via the LEM method to be described in the following sections.

The numerical results obtained from the application of the LEM algorithm are validated against experimental measurements in the round turbulent jet formed at the exhaust of a Garrett 30 – 67 gas turbine engine. For the

* Tel:+4021.434.01.79; Fax:+4021.434.02.41
E-mail address: cristian.carlanescu@comoti.ro ;*URL*: www.comoti.ro

Arclength continuation for implicitly defined curves

Stefan-Gicu Cruceanu*

"Gheorghe Mihoc - Caius Iacob" Institute of Mathematical Statistics and Applied Mathematics, Department of Applied Mathematics, Romanian Academy, Calea 13 Septembrie No.13, Sector 5, 050711 Bucharest, ROMANIA, P.O. BOX 1-24, Phone/Fax (4021)3182439.

Abstract: Continuation methods are addressed in the past few years by a number of surveys and monographs, including [4][11][12][13][14][15], and some software packages have been developed in the context of ordinary and partial differential equations for performing numerical continuation. In most situations arising from the discretization of systems of parametrized differential equations, homotopy continuation occurs for real solutions and has largely been used just once to obtain an initial guess rather than as the major part of the computation.

More recently, arclength continuation has been used to study curves that are implicitly defined by equations $H(t,z)=0$, where $H:[0,1]\times \mathbb{C}^n \to \mathbb{C}^n$ is an analytic map in the z-variables and smooth in the t-variable. This study was started due to the application of homotopy continuation methods to the problem of finding finite dimensional approximations to *all* of the solutions of a class of two point boundary value problems [1] and to finding finite dimensional approximations to *all* of the solutions of a class of semi-linear elliptic boundary value problems [2][6].

2000 Mathematics Subject Classification: Primary, 65H20; Secondary 14Q05, 90C30, 65H10, 65P30.

Keywords: arclength continuation, nonlinear system, bifurcation, turning point, homotopy, solution branch.

1. Introduction

Continuation and homotopy methods have and continue to serve as very powerful and useful tools in mathematics, especially when solving nonlinear system of equations. They have been successfully used, for example, to improve convergence properties of iterative methods when an adequate starting value was not available.

Let's suppose we are looking for a solution to a system of n nonlinear equations in n variables, say

$$F(x)=0, \tag{1}$$

where $F:\mathbb{R}^n \to \mathbb{R}^n$ is a smooth mapping. If a good approximation x_0 of a zero point $\tilde{x}$ of F is available, then one can use a Newton-type algorithm to calculate $\tilde{x}$. Such an algorithm is described by an iteration formula as

$$x_{j+1} := x_j - A_j^{-1}F(x_j), \qquad j=0,1,\dots \tag{2}$$

where A_j represents some reasonable approximation of the Jacobian $F'(x_j)$. But, in general, we find ourselves in the contrary situation when none or very little a priori knowledge concerning zero points of F is known. If applied in such case, the iteration (2) will often fail because poor starting values are likely to be chosen. One remedy for this is to use a homotopy method which basically consists of the following:

1. defining a homotopy $H:[0,1]\times \mathbb{R}^n \mapsto \mathbb{R}^n$ such that

$$H(1,x)=G(x), \qquad H(0,x)=F(x), \tag{3}$$

where $G:\mathbb{R}^n \mapsto \mathbb{R}^n$ is a smooth map (usually a trivial one) having knowing zero points, and H is also smooth;

2. tracing an implicitly defined curve $c(s)\in H^{-1}(0)$ from a starting point $(1,x_1)$ to a solution point $(0,\tilde{x})$, where x_1 is one of the known zero points of G. If this succeeds, then a zero point $\tilde{x}$ of F is obtained.

Two examples of such homotopies are

- the convex homotopy $H(t,x):=tG(x)-(1-t)F(x)$,
- the global homotopy $H(t,x):=F(x)-tF(x_1)$.

A first problem that arises in the reader's mind relates to the conditions in which a curve $c(s)\in H^{-1}(0)$ with

* Tel.: +40 21 318 81 06, int.: 3416; fax: +40 21 318 24 39.
E-mail address: gcruceanu@yahoo.com; *URL:* http://www.ima.ro

A numerical analysis of combustion process in an axisymmetric chamber

Alexandru Dumitrache*♣, Florin Frunzulica*,**, Horia Dumitrescu*
*Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy,**Faculty of Aerospace Engineering, University POLITEHNICA of Bucharest*

Abstract: Combustion phenomenon is one of the most important problems involved in different industries, such as gas turbines, combustion chamber, melting of metals, etc. In this paper, combustion process of methane downstream of an axisymmetric sudden expansion in a circular pipe with a constant wall temperature has been studied. The conservation equations of mass, momentum, energy, and species as well as additional equations due to turbulence modeling have been numerically solved. The standard k-ε model and eddy dissipation combustion model has been used to simulate the turbulence and combustion phenomenon, respectively. Properties of gaseous mixture have been computed using the ideal gas equation of state. The governing equations have been discretized using a finite volume approach and power law scheme and the resulting set of algebraic equations has been solved simultaneously using the SIMPLER algorithm on a non-uniform staggered grid system. The numerical results have been compared with the other's numerical results and experimental data.
2000 Mathematics Subject Classification: 76F25, 74S10
Key words: Combustion chamber, CFD, methane combustion, turbulent flow.

I. Introduction

The design of combustion chambers is of great importance. The geometry of these chambers may change gradually or suddenly. When the fluid path is changed abruptly, flow parameters, flow characteristics and the heat transfer rate are altered consequently. On the other hand, turbulent flows through axisymmetric sudden expansions are influenced by many parameters. Among these are inlet geometry, inlet flow Reynolds number, expansion ratio, step height, inlet turbulence intensity and inlet boundary condition. The interaction between turbulence and combustion is very important in the design of combustion chamber. Combustion phenomenon and the important design parameters are governed by this interaction.

Numerical simulation is becoming a powerful means in understanding combustion process and designing or optimizing combustion systems. The mathematical modeling of turbulent combustion has been outlined by Magnussen et al. [1]. They have developed the eddy dissipation concept for modeling of combustion. Turbulent non-reacting flows have been briefly studied by Ramous [2]. He, later, developed a mathematical model to study turbulent, confined, swirling flows under reacting non-premixed condition [3]. It is observed that the dimensions of the recirculation zone are larger for non-premixed reacting flows than for incompressible conditions. This seems to be caused by the heat released from the chemical reaction, which affects the density, centrifugal forces and radial pressure gradient.

Two fast-chemistry models, the eddy dissipation concept (EDC) and the conserved scalar (CS) approach, have been analyzed [4]. In this work turbulence is simulated by three types of k-ε model. The equations are discretisized using a hybrid scheme and the SIMPLE algorithm is employed to solve the resulting algebraic equations.

Ohtsuka has performed the numerical study of reacting and non-reacting flows using the Reynolds stress differential [5]. He has modeled mixing of air and helium and also the combustion of propane.

In this work, combustion of methane in turbulent flow is studied. It is assumed that the fuel and air enter the cylindrical chamber with abrupt expansion as confined coaxial jets.

♣ Corresponding author: phone +40.21.318.81.06/int. 3405
E-mail address: alex_dumitrache@yahoo.com, *URL*: http://www.ima.ro

Unsteady free-wake vortex particle model for horizontal axis wind turbines

Florin Frunzulică[1,2], Alexandru Dumitrache[2], Horia Dumitrescu[2], Vladimir Cardoş[2]
[1]*Faculty of Aerospace Engineering, University POLITEHNICA of Bucharest,* [2]*"Gheorghe Mihoc – Caius Iacob" Institute of Mathematical Statistics and Applied Mathematics, Bucharest, Romania*

Abstract: In the design of horizontal axis wind turbines (HAWT) one problem is to determine the aeroelastic behaviour of the rotor blades for the various wind inflow conditions. A step in this process is to predict with accuracy the aerodynamic loads on the blades. The Vortex Lattice Method (VLM) provides a transparent investigation concerning the role of various physical parameters which influence the aerodynamic problem. A major problem encountered in the numerical simulation of the three-dimensional incompressible inviscid flows is the expensive computation effort to specify the wake location and its influence. In this paper we present a method for the calculation of the non-uniform induced downwash of a HAWT rotor using the vortex ring model for the lifting surface coupled with an unsteady free-wake vortex particle model. Comparative studies between results obtained with different models of wake for a generic HAWT are presented.
2000 Mathematics Subject Classification: 45K05
Key words: HAWT; vortex lattice method; free-wake; vortex particle.

1. Introduction

For rotor computations, the blades element momentum methods are easily understandable and applicable, using minimum computation requirements. Anyhow, there are cases when these methods do not provide the desired precision. The design interest cases include asymmetric aerodynamic unsteady flow (especially the dynamic effects of incident flow). An evident alternative is the detailed computation of the induced velocities' field, based on the wake vortex distribution [1][2]. The vortex methods are interesting, even while requiring significant computation resources, due to their possibility to observe the vortex system's main structure. In the following sections, we will present a calculation algorithm where the near-wake strip elements are transformed into vortex particles and become part of the far-wake. Integrating the vorticity of each near-wake dipole element produces a vortex particle. The new vortex particles became part of the far wake which evolves prior to the next time step using a Lagrangean description of the flow .

2. Working Hypothesis

The working fluid is a continuous barotropic ideal fluid which fills an unlimited, simple, contiguous domain τ'. In τ the flow is adiabatic and non-rotational, where $\tau=\tau'-\tau_R$. τ_R is the rotational flow domain (i.e. for a blade this is represented by the solid blade surface and the wake flow- field domain).

The blades have a rotational motion related to the reference system attached to the airflow speed direction. We suppose that the blades are rigid body.

In the calculation points (control points) the velocity is given by sum of three components: the cinematic velocity corresponding to blade motion, the induced velocities $\mathbf{V}_{ind,P}$ and the wake induced velocities $\mathbf{V}_{w,P}$:

$$\mathbf{V}_P = \mathbf{V}_{k,P}(t) + \mathbf{V}_{ind,P} + \mathbf{V}_{w,P} \tag{1}$$

Another assumption is that the generated free wake is attached to the blade.

The time steps are chosen so that vortexes generated by the trailing/leading edge should not transported on distance greater than the smallest panel dimension.

[1] Corresponding author: phone +40.21.402.39.67, fax +40.21.318.10.07
E-mail address: ffrunzi@yahoo.com, *URL*: http://www.aero.pub.ro

Self-propulsion of thin profiles; an analytic approach

Stelian Grădinaru*

* *Spiru Haret University of Bucharest, Romania*
stelian_gradinaru@yahoo.com

Abstract: The paper deals with the incompressible flow past oscillating thin profiles. In the frame work of the linearized theory, the pressure jump over the oscillatory profile is the solution of a hypersingular integral equation. Using an asymptotic expansion of the kernel with respect to the frequency of oscillation (considered as a small parameter) and keeping the leading terms, a simplified form of the integral equation has been found, known in the literature as Possio's equation. General solution is obtained in terms of analytical approach and has an integral representation. Some configurations of oscillatory and undulatory thin profiles are considered: the flat plate, the parabolic profile and S-profile. For each case, the aerodynamic coefficients are evaluated. For some certain values of the frequency of oscillation, the average drag coeffcient becomes negative so that the oscillating motion induced the appearance of a propulsive force
2000 Mathematics Subject Classification: 76
Keywords: fluid dynamics; unsteady aerodynamics; oscillatory wing; singular integral equations; thin profile.

1. Introduction

The study of the self-propulsion induced by the oscillatory and undulatory motions of rigid or flexible airfoils represents an actual problem in aerodynamic with many applications in engineering of micro vehicles motion. Many authors have treated in their work computational methods in fluid dynamics investigating this topic. Herein the method of the integral equations is used to the study of incompressible flow past thin profiles performing an oscillatory or undulatory motion as the wing is rigid or flexible respectively. As we know, on the particularly case of steady flows, the theory yields exact solutions when the angle of attack is small and the profile is enough thin such that linearization of the equations of motions makes possible an analytic approach of the problem. In exchange of limitations related to the geometry of the airfoil, the theory of the thin profiles has a rigorous formulation and using the method of the integral equations the pressure jump across the profile is obtained.

Among the scientists have treated the three-dimensional problem of the oscillatory wing, we remind some of them. Dragoş [4] studied the compressible flows past oscillating wings by means of fundamental solutions. Homentcovschi [5] used Laplace – Fourier transforms to find the fundamental solution of the linear system and then, the general integral equation relating the pressure jump and the downwash distributions for the unsteady flow past lifting surface on the hypothesis that surface is moving on the cylinder. Later, Carabineanu [1] has deduced the fundamental solution of the linear Euler system and applies theory of distributions to find the singular integral equation of oscillating wing Carabineanu [2] also used numerical methods to compare them to the exact solutions.

Many other papers develop numerical methods to study this topic and come to fill the emptiness due to the difficulties of the singularities of the kernel.

Our paper gives a chance to the analytical methods to find the exact solution for some particular profiles. The used method herein is following Carabineanu [1] with slightly modifications.

Solution of the 2D linear system is found with aid of distributions. Imposing sliding condition for the downwash, a hypersingular integral equation is obtained. We recognize here Possio's equation. The aim of this work is to formulate an analytical method which solves the singular integral equation for the pressure jump in the case of the thin profile subjected to the oscillatory or undulatory motions. Reducing of the singularity is possible once we make a simple and real supposition: the wing is subjected to a low frequency of oscillation, ω

* *E-mail address:* stelian_gradinaru@yahoo.com

The effect of saturation on the longitudinal command in the approaching phase of flight for landing

Ionel Cristinel Iorga[1]

* *University of Craiova, Alexandru Ioan Cuza Str. 13, 200585 Craiova, Romania*

Abstract: In this article we study the effect of saturation on the longitudinal command in the approaching phase of flight, for landing. Two models are used, in order to do this: (α, q, δ_e) ((5),(4)) and $(V, \alpha, q, \theta, \delta_e)$ ((6),(4)). The models use different slopes of flight and pilot gains. We study the stability of the proposed systems.
2000 Mathematics Subjects Classification: 0350; 65C20; 91B47; 34A34; 37M05; 37N10; 39A11
Keywords: longitudinal flight model; flight control system; stability; simulation

1. *Introduction*

In literature among classic results as the one of Hess [2], we note the results of Balint et all. [3].
We start from the following system (from [1]):

$$\begin{cases} \dot{\alpha} = z_\alpha \alpha + z_{\delta_e} \delta_e + \dfrac{g}{V_0}\cos(\theta) + q \\ \dot{q} = \overline{m}_\alpha \alpha + \overline{m}_q q + \overline{m}_{\delta_e} \delta_e + \dfrac{g}{V_0}\left(m_{\dot{\alpha}} \cos(\theta) - \overline{a}\sin(\theta)\right) - \dfrac{1}{a}\alpha q \\ \dot{\theta} = q \\ \dot{\delta}_e = \omega_0 \psi(\sigma) \end{cases} \tag{1}$$

Remark 1. z_α = -0.7985, z_{δ_e} = -0.2603, $\overline{m}_\alpha$ = -6.4774, $\overline{m}_q$ = -0.6957, $\overline{m}_{\delta_e}$ = -8.2667, $m_{\dot{\alpha}}$ = -0.162,

$\omega_0 = 20\ \frac{rad}{s}$, $e_L = \dfrac{V_0}{\omega_0}$, $\overline{a} = 1.423$, $g = 9.81\dfrac{m}{s^2}$

$$\psi(\sigma) = \begin{cases} \sigma, if\ |\sigma| \le e_L \\ e_L \operatorname{sgn}\sigma, if\ |\sigma| > e_L \end{cases} \tag{2}$$

$$\sigma = \delta_c + \delta_e \tag{3}$$

Remark 2. δ_c, from the equation (3), is a variable which perturb the dynamical equilibrium of the aircraft. This variable is small.

Remark 3. V_0 is the aircraft speed (84.5 m/s) and σ - additive input value. e_L is the limit for saturation.

Remark 4. The global gains are: k_α = 0.401, k_q = 1.284, k_p = 0.521

$$\delta_e = k_\alpha \alpha + k_q q + k_p \theta \tag{4}$$

[1]* Tel: +40 720378788; fax: +40 249515301
E-mail address: ioneliorga@yahoo.com ; *URL:* http://www.ucv.ro

A numerical study of laminar flow past two circulating spheres in tandem

Gheorghe Juncu*, Aurelian Nicola** and Constantin Popa**
*Politehnica University Bucharest,Catedra Inginerie Chimica, Polizu 1,Bucharest 011061, Romania,
**Ovidius University, Department of Mathematics, Bl. Mamaia 124, Constanta 900527, Romania,

Abstract: Numerical methods are used to investigate the steady, axisymmetric, viscous flow past two equal size fluid spheres in tandem. The vorticity – stream function formula[1]tion of the Navier – Stokes equations was used. The momentum balance equations were solved numerically in bispherical (for flow outside the spheres) / spherical (for flow inside the spheres) coordinates systems by a finite difference method. The computations were focused on the influence of the spheres spacing on the drag coefficients for Re ≤ 1 and three fluid – fluid systems: gas-bubbles in liquids, liquid drops in a different immiscible liquids and liquid drops in gases.
2000 Mathematics Subject Classification: 76T10, 76T15, 76M20
Keywords: Laminar flow, two spheres in tandem, bispherical coordinates, Navier-Stokes equations, stream function, vorticity, drag coefficient.

1. Introduction

The hydrodynamic interaction between two spheres moving in a viscous fluid is a widespread phenomenon in many natural and industrial processes. The importance of this phenomenon is reflected by the large number of articles published over the years. Spheres placed side – by – side and in tandem formation are the two most frequently investigated dual spheres arrangement. The case analysed in the present work is the flow past two gas bubbles or liquid droplets in tandem (i.e. spheres with mobile interface). For this reason, only the studies directly pertinent to this problem are reviewed here. A recent article for flow past two fixed rigid spheres in tandem is Prahl et al. [1].

For creeping (Stokes) flow, analytical (exact or approximate / asymptotic) solutions are presented in [2 – 11]. These studies focused on describing hydrodynamic interactions between an arbitrarily – oriented pair of spherical drops that are either moving with prescribed velocities, or under the action of an external force such as gravity, or freely – suspended in a linear flow. Pigeonneau and Feuillebois [12] reviewed the most important results obtained for Stokes flow. An exact, analytical solution to the creeping flow problem of two stationary spherical drops that are touching in a head-on collision, was derived in [13] based on the tangent sphere coordinates.

The steady, three – dimensional incompressible laminar flow of a Newtonian fluid past two identical (solid and liquid) spheres held fixed, with the line connecting the spheres centres normal to a uniform stream (side by side arrangement), was investigated by Kim et al. [14]. The analysis of the flow field past two liquid spheres was performed for: (1) dimensionless distance from the sphere centre to the symmetry plane between the spheres in the range [1.5, 25]; (2) viscosity and density ratios (internal fluid to external fluid) of 25 and 300, respectively; (3) Reynolds numbers (external flow) 50, 100 and 150. These values are typical of liquid – hydrocarbon fuel in a high – temperature, high – pressure surrounding gas generally encountered in gas turbine combustors.

The aim of this work is to solve numerically the Navier - Stokes equations for the steady, laminar, incompressible flow past two circulating spheres of equal diameters in tandem arrangement. The vorticity – stream function formulation of the Navier - Stokes equations in bispherical / spherical coordinates was used. Different spheres spacing were considered for the sphere Reynolds number varying between 10^{-4} and 1. Three systems were taken into consideration: gas-bubbles in liquids, liquid drops in a different immiscible liquid and liquid drops in gases. To our knowledge, this problem was not investigated until now.

The references are listed at the end of the paper in order that they are cited in the text.

*Tel: +40 (0)21 345 0596; fax: +40 (0)21 312 6879
E-mail address: juncugh@netscape.net; anicola@univ-ovidius.ro; cpopa@univ-ovidius.ro;

Applications of visualization methods in aeronautics and civil engineering

Ion Lăncrănjan[1*], Ionuț Brînză[**]

[*] *Advanced Study Centre – National Institute for Aerospace Research "Elie Carafoli", ilancranjan@incas.ro, B[dul]. Iuliu Maniu 220, Sector 6, 061126, Bucuresti*

[**] *National Institute for Aerospace Research "Elie Carafoli", INCAS, Experimental Aerodynamics Laboratory, ibrinza@incas.ro, B[dul]. Iuliu Maniu 220, Sector 6, 061126, Bucuresti*

Abstract: Flow visualization is an important topic in scientific investigation of phenomena and has been the subject of active research for many years. Typically, data originates from numerical simulations, such as those of computational fluid dynamics (CFD), and need to be analyzed by means of visualization in order to gain an understanding of the flow. Preliminary experimental results obtained in developing a visualization technique for non-invasive analysis of air flow inside INCAS subsonic wind tunnel and its appendages are presented. The visualization technique is based on using a green light sheet generated by a continuous wave (cw) longitudinally diode pumped and frequency doubled Nd:YAG laser. The output laser beam is expanded on one direction and collimated on perpendicular on it one. The system is tailored to the requirements of qualitative analysis and vortex tracking requirements inside the INCAS 2.5m x 2.0m subsonic wind tunnel test section, for measurements performed on aircraft models. Also, the developed laser visualization techniques are used for non-invasive air flow field analysis into environmental facilities settling room (air flow calming area). Future quantitative analysis is enabled using special image processing tools upon movies and pictures obtained during this laser visualization experiment. The basic experimental layout in the wind tunnel takes advantage of information obtained from the investigation of various aircraft models using the developed visualization technique. The results are important for prevention of biological/ chemical disasters such as spreading of extremely toxic pollutants due to wind. Numerical simulations of wind flow and experimental visualization results are compared. A good agreement between these results is observed.

2000 Mathematics Subject Classification: 37N20

Keywords: air flow, tufts, oil visualization, laser light sheet, experimental aerodynamics, civil buildings

Tuft flow visualization

For this experiment, cotton threads segments have been attached on the upper part of the right wing. The length of the threads was of 45 mm each, the active length being of 35 and the rest of 10 mm used for fixing on the wing.

The attachment was made with small rectangular pieces 10x14 mm of adhesive tape. On the chord direction, the tufts were distributed with a fixed step of 10% of chord length and span wise, the threads were placed with a 45 mm distance between them.

This placing of the tufts generated a rectangular distribution matrix. As a future work, we intend to use two other configurations for this matrix: the double column and zigzag distribution.

[1] Tel.: +40 021 434 0083; fax: +40 021 434 0082
E-mail address: ilancranjan@incas.ro; *URL:* http://www.incas.ro

Finite element modeling of blood flow in an arterial bifurcation

Alexandru M. Morega*, Alin Dobre**, Mihaela Morega***, Daniel Mocanu****
*University POLITEHNICA of Bucharest, Romania, Faculty of Electrical Engineering; Institute of Statistical Mathematics and Applied Mathematics "Gh. Mihoc – C. Iacob", Romanian Academy amm@iem.pub.ro
** University POLITEHNICA of Bucharest, Romania, Faculty of Electrical Engineering alin_dobre@yahoo.com
*** University POLITEHNICA of Bucharest, Romania, Faculty of Electrical Engineering mihaela@iem.pub.ro
**** University POLITEHNICA of Bucharest, Romania, Faculty of Electrical Engineering moccad@yahoo.com

Abstract: It is well known that hemodynamic factors related to blood flow are strongly correlated with atherosclerotic arterial disease. In this paper, we present a finite element model of the arterial blood flow that may be used for the study of atherogenesis, pre-surgical planning or drug targeting. The computational model includes segments of the ascending aorta, brachiocephalic and right common carotid. The vessels were segmented out of contrast-enchanced magnetic resonance images of the human arterial system and discretized using Simpleware software package. The geometry and the finite element mesh thus obtained by utilizing Simpleware and SolidWorks software packages were exported to COMSOL for numerical analysis. Simulation results are presented for various arterial bifurcations under steady-state and pulsatile flow conditions.
2000 Mathematics Subject Classification: 76D05
Keywords: blood flow; 3D reconstruction; hemodynamic; finite element analysis.

1. Introduction

Atherosclerosis is a syndrome affecting the arterial blood vessels. This chronic disease is characterized by the thickening of the vessel wall due to accumulation of fatty materials known as atheromatous plaque [1,2]. The build-up of plaque causes partial or total blockage of the blood flow and may lead to tissue ischemia and infarction if not treated. There are several processes at the cellular level that play an important role in atherogenesis. However, it has been widely observed that plaque deposits are strongly correlated with complex patterns of arterial blood flow. The major regulation of the blood vessel occurs by the action of hemodynamic shear stresses on the vascular endothelium [3]. Thus, the mechanical forces related to blood flow are key to the vascular tone control, the morphology of the arterial system and atherogenesis.

Many studies have been devoted to understanding the blood flow hemodynamics and the associated mass transfer processes [5,6,7,12]. Recent advances in medical imaging technologies and computer processor speed have made possible the investigation of arterial flow using image-based, anatomically realistic computational models. Conventional one-, three-dimensional, and hybrid numerical schemes were employed to perform hemodynamic analysis under either steady or pulsatile flow conditions, on several anatomically accurate models.

In this paper, we present a finite element model and simulation results for blood flow simulation. The computer model was constructed by using magnetic resonance images of the arterial system. Numerical results are presented for various arterial bifurcations under steady-state and pulsatile flow conditions.

2. Medical imaging based geometry

The 3D computational model includes segments of the ascending aorta, brachiocephalic and right common carotid (Fig. 1). The vessels were segmented out of contrast-enchanced magnetic resonance images of the human arterial system and discretized using Simpleware software package. The geometry and the finite element mesh thus obtained by utilizing Simpleware and SolidWorks software packages [8,11] were exported to COMSOL [9] for finite element analysis.

* Tel: (+40 21) 4029153,4029149,4029125; Fax: (+40 21) 3181016
E-mail address: amm@iem.pub.ro ; *URL:* http://www.iem.pub.ro

Noise metric for high-lift-low-noise concept validation

Cătălin Nae*
INCAS – National Institute for Aerospace Research "Elie Carafoli" , Bucharest, Romania

Abstract: Present analysis proposes a new noise metric and a global procedure that may be used for optimization problems involving aerodynamic noise from a clean wing. This metric is based on a classical trailing edge noise theory as the starting point where we include characteristic velocity and length scales that are obtained from three-dimensional URANS simulations with k-eps turbulence model. Noise metric validation was performed with a test cases that was selected from a two dimensional NACA experimental database. The agreement between the experiment and the predictions obtained with the new noise metric was very good at various speeds, angles of attack, and Reynolds numbers, which showed that the noise metric is capable of capturing the variations in the trailing edge noise as a relative noise measure when different flow conditions and parameters are changed. Direct application is the optimization of the L1T2 high lift configuration. This is part of a on-going project based on the theoretical development of High-Lift-Low-Noise (HiLoN) concept to be used for a next generation of high lift systems for green aircraft, as considered in EU FP7 JTI Clean Sky.

Key words: aeroacoustics, noise, CFD, wind tunnel testing

1. Introduction

The airframe noise is an important noise source mainly for large aircraft in its landing and take off configuration. Also, the level of noise produced just by the passage of an airplane through the air, especially in its landing configuration, may only be a few decibels below the level of noise radiated from the engines. As a consequence, with respect to new designs for green aircraft, the main interest is for landing/take-off configurations and steep descent/climb maneuvers with reduced noise emission. Since we traditionally associate noise with the size and weight of the flying body, we can imagine an optimization process where one is interested to identify the optimum shape with respect to noise emissions giving the requested lift (here referred as High-Lift-Low-Noise concept)

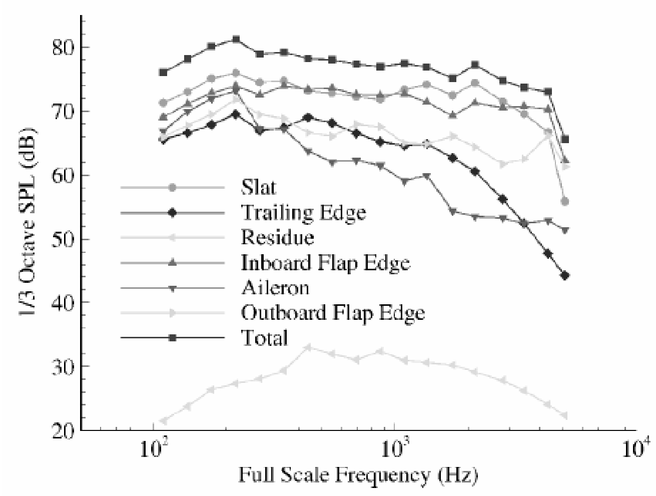

Figure 1 - Source of airframe noise and reference values

In order to have an estimate of the noise sources localization and reference values in the case of a landing configuration, we present results from a specific analysis in Figure 1 [1]. Here we emphasize that one might

* Tel. +40.745.780.140 , fax. +40.21.434.0082
E-mail address: cnae@incas.ro; *URL:* http://www.incas.ro

Analysis of aerodynamic field and noise from a ducted wind turbine

Mihai Leonida Niculescu*, Mircea Dan Ionescu**, Marina Andrei**, Cristian Cercel**
*), **) *National Research & Development Institute for Gas Turbines COMOTI,*
220 D Iuliu Maniu Avenue, 061126 Bucharest 6, Romania, P.O. 76, P.O.B. 174
e-mail: mihai.niculescu@comoti.ro, dan.ionescu@comoti.ro, marina.andrei@comoti.ro, cristian.cercel@comoti.ro

Abstract: The wind energy is deemed as one of the most durable energetic variants of the future because the wind resources are immense. For example, one of the most comprehensive studies showed that the retrievable wind power on land and near-shore is about 72 TW, which represents 5 times the world's current energy use. As the wind energy market matures and becomes more competitive, it is very important to realize new wind turbines with an increased efficiency, even for low wind speeds.
Keywords: ducted wind turbine; CFD; noise

1. Introduction

One predicts that the small wind turbine will play a vital role in the urban environment. Unfortunately, nowadays, the noise emissions from small wind turbine represent one of the main obstacles to widespread the use in populated zones. Furthermore, the energetic efficiency of these wind turbines has to be high even at low and medium wind velocities because, usually the cities are not windy places.

From these reasons, we consider that it is useful to study both the aerodynamics and noise from a small-ducted wind turbine in order to find out the ways to increase the aerodynamic efficiency and to minimize the noise. The aerodynamic field is study with the finite volume method using the Reynolds averaged Navier-Stokes (RANS) equations while the acoustic field is study with the broadband noise source models implemented in FLUENT.

2. Nomenclature

- a speed of sound (m/s)
- e internal energy (J/kg)
- f_e external acceleration (m/s^2)
- F_x, F_y, F_z vectors of convective components of flux
- G_x, G_y, G_z vectors of diffusive components of flux
- I rothalpy (m^2/s^2)
- k turbulent kinetic energy (m^2/s^2)
- p static pressure (Pa)
- P total pressure (Pa)
- r radius (m)
- R gas constant (J/(kg·K))
- S vector of source term
- T static temperature (K)
- t time (s)
- u, v, w Cartesian components of velocity (m/s)
- V absolute velocity (m/s)
- W relative velocity (m/s)
- y^+ dimensionless wall distance
- Δt physical timescale (s)
- ε rate of dissipation of turbulence kinetic energy (m^2/s^3)
- γ specific heat ratio

* Tel: +4021 434 01 98; fax: +4021 434 02 41
E-mail address: mihai.niculescu@comoti.ro; *URL:* http://www.comoti.ro

A numerical study of the heat transfer by the aluminum electrolysis cell shell

Marin V. Petre[*], Alexandru M. Morega[**,***]

[*] *ALRO Slatina Plant, No.116, Pitesti Street, 230048, Slatina, Romania*

[**] *Faculty of Electrical Engineering, University POLITEHNICA of Bucharest, Bucharest, 060042, Romania.*

[***] *"Gheorghe Mihoc – Caius Iacob" Institute of Mathematical Statistics and Applied Mathematics, Romanian Academy, Bucharest, Romania*

Abstract: This paper presents a study of the heat transfer and airflow for a new shell design of an electrolysis cell at ALRO Slatina aluminum electrolysis plant. To validate this new design, experiments were performed on a test group of 15 cells, and specialized equipment was used to measure the temperature of the cell shell wall. To better understand the heat transfer mechanisms and to provide for thermal design optimization solutions, a mathematical model and its numerical implementation (finite element, FEM) are formulated. Numerical simulation was performed to investigate the natural convection heat transfer and the airflow by the cell shell, and the results are compared with the available experimental data.

2000 Mathematics Subject Classification: 76D05
Keywords: aluminum electrolysis, heat transfer, numerical simulation, finite element analysis.

1. Introduction

Aluminum electrolysis by the Hall-Heroult process requests a lot of power therefore improving its efficiency is a matter concern. Reduction line currents for older technologies are typically 100 to 200 kA, whereas the new plants operate at about 350 kA. Trials have been reported with 500 kA cells [1]-[3]. By increasing the electrical current the heat generated within the cell increases too, therefore higher excessive occurs. The cell shell participates in the heat transfer exhausted from the cell to the environmental air. With ever increasing power levels, its capacity to dissipate the heat has to be adjusted consistently to the expected thermal balance of the pot.

One of the solutions of thermal cell design optimization consists of modifying the standard pot into a suspended cell, which provides for better ventilation, especially at the bottom, and most of the electrolysis halls constructed recently are using suspended cells. For older electrolysis halls with pots residing directly on the ground, as is the case of ALRO plant, ventilation channels are provided under the cells.

A significant contribution to the cell design optimization may be brought by the numerical simulations – of different degrees of resolution and complexity – of the conjugated electro-thermal, electromagnetic, chemical, and structural processes that occur within the cell under the working conditions. For instance, a mathematical model of the electrolysis cell is reported by [4]; the heat transfer problem by the lateral wall of the cell shell was formulated and solved numerically by the finite element technique in [5] – to name only a few.

This paper extends previous experimental and numerical simulation studies [14] for the heat transfer at the case wall of a suspended cell. Experimental data was input to the models aiming at providing for accurate working conditions, and thus better depicting the heat transfer mechanisms from the cell shell to the ambient. The results that were obtained are in fairly good agreement with experimental data.

2. The electrolysis cell shell model

The initial design of a standard cell's metallic shell that is sketched in Fig. 1, *a* [2] comprises several major components: (*a*) a demountable superior plate; two belts on the lateral wall – one of them is *U*-shaped (b) at the metal bath interface level, and the other one is *I*-shaped (c), it strengthens the shell at the sub-cathode level – interconnected through vertical belts (d). Several pillars, placed between the walls of two adjacent cells, restrict the bending of the shells walls.

[*] Tel: (+40 21) 4029153,4029149,4029125; Fax: (+40 21) 3181016
E-mail address: amm@iem.pub.ro ; *URL*: http://www.iem.pub.ro

Mathematical modelling of a giant unilamellar vesicle under positive osmotic pressure

Dumitru Popescu[*], Alin Gabriel Popescu[**], Ecaterina Mărieş[***]

[*]*Department of Mathematical Modelling in Life Sciences, Institute of Mathematical Statistics and Applied Mathematics, Romanian Academy of Science, Calea 13 Septembrie, nr. 13, sector 5, Bucharest-050911, Romania*
[**]*Department of Computer Sciences, IT CORE SRL, Str. Garoafei, nr.10, sector 5, Bucharest-051235, Romania*
[***]*Department of Neurobiology and Biophysics, Faculty of Biology, University of Bucharest, Spl. Independentei nr. 91–95, Bucharest, Romania*

Abstract: Under certain conditions, the membrane tension of a giant unilamellar lipidic vesicle is essentially nil. Consider such vesicle containing an aqueous solution of a substance which is not permeating the vesicle membrane. This vesicle is immersed inside a large box filled with water. Due to osmotic pressure, water inflows into the vesicle through the lipidic membrane, stretches vesicle, increasing the membrane tension until the membrane responds by the sudden formation of only a single transmembrane pore at a time. The osmotic stress and the transmembrane pore appearance are the two processes which determine the vesicle to experience a periodic process with its swelling and relaxing stages in each cycle. In this paper, we have obtained the differential equations which describe the both stages of o cycle. The differential equation for the swelling process has an analytical solution which describes the dependence of the swelling time on the vesicle radius. The relaxing stage is described by a system of three differential equations. Following the numerical integration of this system we have obtained: the time dependence of the vesicle radius, pore radius and internal substance concentration during the relaxing stage.

2000 Mathematics Subject Classification: 92
Keywords: osmotic stress; transmembrane pore; pulsatory liposome; bioengine

1. Introduction

The pore appearance in lipid bilayers following some controlled processes may be an adequate and interesting way for transmembrane transport.

There are a lot of theoretical and experimental papers, about the pore formation in plane lipid bilayers, but there aren't many papers about the pore opening in the lipid vesicle [1, 2, 15 – 17].

The pore appearance may be influenced by structural defects as thickness fluctuations[4 – 6, 10] or cluster existence [7, 8].

Some pores, named stochastic pores, can appear due to structural and dynamic properties of lipid bilayer [4-8,10], but others may be favored by mechanical tension induced by different ways [11, 12]. Recently, a sequence of 30–40 pores was observed in the same giant vesicle, a pore at a time, which can appear in vesicles stretched by optical induced mechanical tension [1, 2, 15].

There are two very interesting biotechnological applications which request the increase of membrane permeability: gene therapy and targeted special substances delivery. In the first one, the transport of DNA fragments through cellular and nuclear membranes is requested [16]. The second application uses special substances molecules encapsulated in vesicles, which have to be transported to a target place [17].

Having reached that point, one supposes that the liposome discharges its content in the external medium by its breakdown.

In the last our three papers we have written about how a lipid vesicle has to release the drug molecules, in a well-controlled fashion [12–14]. It must work as a pulsatory liposome whom energy is supplied by the concentration gradient across membrane of a impermeant solute.

[*] Corresponding author. Tel./Fax: +40213182439
E-mail address: popescu1947@yahoo.com; *URL:* http://www.ima.ro

Similar solution for a Coanda flow and applications to directed synthetic jets

Octavian Preotu*♣, Alexandru Dumitrache**, Horia Dumitrescu**

*Univesity of Craiova, Faculty of Electrotechnics, Blvd Decebal , No. 5, Craiova, Romania,
** Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy

Abstract: The presented work deals with the steady, incompressible, two-dimensional flow that develops on the surface of a circular cylinder, like a thin jet sheet, called Coanda flow. Due to the presence of the cylindrical surface the entrainment of the jet is inhibated on the side of the curved surface. This entrained fluid must then accelerate over the Coanda surface producing a local pressure region, which results in a pressure gradient perpendicular to the jet centerline. It is shown that the Coanda effect can be approximated by similar solutions both in laminar and turbulent flow. An application based on the Coanda flow interacting with directed synthetic jets is also described.
2000 Mathematics Subject Classification: 76B75, 76B10
Key words: Coanda effect, synthetic jet, similar flows.

I. *Introduction*

Jets are frequently observed to adhere to and flow round nearby solid boundaries. This general class of phenomena, which may be observed in both liquid and gaseous jets, has become known as Coanda effect. The flows deflected by a curved surface caused great interest in recent years. Major interest is due to the possibility of using this effect to aircrafts with short takeoff and landing for fluidic vectoring.

This work deals with the steady two-dimensional, laminar and turbulent flow of an incompressible fluid that develops like a jet-sheet on a cylinder surface, i.e. a Coanda flow. It shows that this flow can be approximated well enough with similar solutions for both laminar and turbulent regime. Basically it is use of Falkner-Skan type transformations of the momentum equations that can be reduced to one ordinary differential equation (ODE). These solutions are presented in the paper for both laminar and turbulent flow. The results are given in the form of analytical expressions for mass flow, thrust and jet-sheet thickness depending on the angle of deviation.

It is also considered the possibility of the thrust augmentation due to fluid entrainment of the jet flow. Thrust vectoring of aircraft that is the key technology for current and future air vehicles, can be achieved by utilizing Coanda effect to alter the angle of the primary jet from an engine exhaust nozzle. Furthermore the increased entrainment by the Coanda surface coupled with the primary jet fluid can augment the thrust [5].

The problem considered here is only a crude approximation of the physical phenomenon. However, we believe that the singular solutions presented can still help as a guide in approaching the problem more accurately further.

II. *Mathematical Model*

Let consider the steady two-dimensional flow of an incompressible fluid develops on a cylindrical surface like a jet-sheet. The boundary-layer type equations are written in a curvilinear coordinate system shown in Figure 1. Assuming that the width of the jet slot is small compared to the curvature radius of the cylinder, R, the boundary-layer approximations can be applied, such the simplified equations of motion are written as:

♣ Corresponding author: phone +40766 228735
E-mail address: opreotu@yahoo.com, *URL:* http://elth.ucv.ro

Optimization and stability issues in simulating the running regimes of the jet engines

Virgil Stanciu*, Irina Carmen Andrei**,
*"POLITEHNICA" University of Bucharest Faculty of Aerospace Engineering "Elie Carafoli" Department of Aerospace Sciences Str. Gh. Polizu, nr. 1-7, sector 1, Bucureşti, 011061, ROMANIA. * Prof. PhD (h.c.), Dean,*
*** PhD, MSc AE, Lecturer* ** ICAndrei28178@gmail.com, irina.ac@aero.pub.ro

Abstract: This paper presents a new approach in jet engine optimization: new constructions with improved performances. Aside the well-known optimization criteria (increasing the thrust and reducing the fuel consumption of a jet engine), the construction is modified such that the turbofan (seen as a twin jet engine) is extended to the triple jet engine, and so far to a more general solution, which is the multiple jet engine. An analytical linearized model represented by the state and output equations, is used to simulate the running regimes of a mixed jet turbofan with afterburner. The test case is the F-100 turbofan. The stability of the jet engine's running regimes at any operating condition is appreciated by the means of the Routh-Hurwitz criterion; therefore, the eigenvalues of the matrix A that appear within the state and output equations had been computed. The interpretation of the results points out the necessity of permanent condition monitoring and optimal control; therefore, modern jet engines have integrated control and monitoring systems.
2000 Mathe matics Subject Classification: 74Pxx; 74P10; 76D55; 76N25; 79Mxx; 49Q10; 35Q93; 37xx; 37M05; 37M99; 35P30; 35Q93; 45C05; 47A75; 47J10; 49xx; 65N25; 93B60.
Keywords: jet engines, optimization, improving the performances and construction, turbofan, multiple jet engine, simulation of running regimes, operating condition, stability, analytical model, state and output equations, Routh-Hurwitz criterion, eigenvalues, optimal control, integrated systems, research, design, aerospace sciences.

1. Introduction

The intent of this paper is to present a study introducing a new optimized construction of jet engine, which features less noise levels and thus being highly recommended for more efficient and environmental friendly air transportation.

In Part A are shown: the progress of the research, the new jet engine concept, i.e. the triple flow jet engine **3FJE**, a comparison of the **3FJE** versus a twin flow jet engine **2FJE** (i.e. a large by-pass turbofan); the advantages of the new **3FJE** are pointed out.

For every jet engine, universal maps of parts can be determined and plotted. With the aid of the Compressor Universal Map CUM (or Compressor System Universal Map CSUM) and the Turbine Universal Map TUM, one can get other characteristic map of the jet engine, such as: the Universal Map of the Jet Engine, the Flight Map, and so on, [9, 11-12].

The Universal Map of the Jet Engine represents the correlation of the thrust F versus specific fuel consumption c_{sp}; which is obtained with the aid of the Operation Line **OL** (also referred as the Working Regimes Line WRL), see Fig. 3.1 in ref. [9], pp. 121.

Twin spool jet engines run safer rather than the single spool jet engines, since the Operation Line is farther from the surge line; in case of triple spool jet engines (as many large by-pass turbofans are, where the fan, LPC and HPC run at different speed regimes) the Operation Line is at farthest from the surge line.

Since the triple flow jet engine 3FJE has at least three spools, it comes out that this construction runs within the stability domain, a consequence of the fact that the Operation Line **OL** is the farthest from the surge line; therefore, the running regimes of the 3FJE are inside the stability domain.

** Tel: +40 21 4023967, +40 723 859352, Fax: +40 21 3181007,
E-mail address: ICAndrei28178@gmail.com , *URL:* http://www.upb.ro, http://www.aero.pub.ro

The influence of saturation relative to the aileron control in the roll motion

Claudia Alice State*

* *University of Craiova, Alexandru Ioan Cuza Str. 13, 200585 Craiova, Romania*

Abstract: In this paper we study the influence of saturation relative to the aileron control in the roll motion (system determined by ((1), (2)) and, for the system described by ((4), (5),(6)) we use also the rudder control). The main goal is to study stability of the proposed systems. For this purpose, two models are highlighted: (p,ϕ,δ_a) (1), (2)) and $(\beta, p, r, \phi, \delta_a, \delta_r)$ ((4), (5),(6)). Systems are analyzed on equilibrium conditions of the longitudinal movement [1].

2000 Mathematics Subjects Classification: 65C20; 37N10; 0350
Keywords: lateral-directional flight model; flight control system; stability; simulation

1 Introduction

The systems use the data's from Straero Institute [2], which further expands some aspects of practical interest of the Admire flight database [3]. In this field, among the classical results of Roskam [5] we can give, as an example, the paper [4].

2 The (p,ϕ,δ_a) system

We start from the differential equation system (from [2]):

$$\begin{cases} \dot{p} = l_p p + l_{\delta_a}\delta_a \\ \dot{\phi} = p \end{cases} \tag{1}$$

where we have:

$$\delta_a = k_p p + k_\phi \phi \tag{2}$$

Remark 1. δ_a is the command for the aileron.
Remark 2. l_p = -19.93, l_{δ_a} = 13.67
Remark 3. $k_p = 0.521, k_\phi = -0.277$ (global gains).

We have the following Figures:

* Tel: +40 720067895; fax: +40 251415902
E-mail address: cldstate@yahoo.com , *URL:* http://www.ucv.ro

Intrinsic analytic study on an isoenergetic flow of a compressible fluid Part 1: "Isentropic" and "zero-work" surfaces in aero-gas dynamics

Richard Şelescu*
Department of Aerodynamics, "Elie Carafoli" National Institute for Aerospace Research – INCAS, 061126, Bucharest, Romania

Abstract: This part studies and clarifies some local phenomena in fluid mechanics. A model of an isoenergetic flow of an inviscid fluid is introduced, in order to establish a simpler form for the general PDE of the velocity potential. It consists in using an intrinsic system of triorthogonal curvilinear coordinates (one of them being tied to the local specific entropy value). The choice of this system (with two coordinate curves lying on the "isentropic" surfaces) enables the treatment of any 3-D flow (rotational, steady and unsteady) as a potential 2-D one, introducing a 2-D velocity "quasi-potential", specific to any isentropic surface. The dependence of the specific entropy on this "quasi-potential" was established. On the above surfaces the streamlines are orthogonal paths of a family of lines of equal velocity "quasi-potential". The model was extended to viscous fluid flows, introducing the "zero-work" surfaces and a new physical quantity – Şelescu's "roto-viscous" **Ş** vector. The new first integrals are similar to D. Bernoulli and D. Bernoulli–Lagrange ones, being obtained by a procedure of eliminating the non-conservative terms in the respective equations. The PDE of the velocity potential, that of the isentropic surfaces, and those of Şelescu's vector lines and zero-work surfaces, are given.
2000 Mathematics Subject Classification: 31 Potential theory; 70 Mechanics of particles and systems; 76 Fluid mechanics; 80 Classical thermodynamics, heat transfer
Keywords: Rotational flows; Steady and unsteady flows; Inviscid and viscous fluids; Compressible fluids; Isentropic surfaces; Şelescu's vector, space curves and zero-work surfaces

1.1. Introduction, nomenclature and the first approach to the new model in aero-gas dynamics

Let us start from the general form of the Euler differential equation of motion (for an inviscid fluid flow):

$$\frac{\partial \mathbf{V}}{\partial t} + \nabla\left(\frac{1}{2}\mathbf{V}^2\right) + \mathbf{\Omega}\times\mathbf{V} = \mathbf{f} - \frac{1}{\rho}\nabla p \quad , \quad \text{with} \quad \nabla = \mathbf{k}_x\frac{\partial}{\partial x} + \mathbf{k}_y\frac{\partial}{\partial y} + \mathbf{k}_z\frac{\partial}{\partial z} \quad \text{– nabla (Hamilton's operator)}$$

(the Gromeko–Lamb form, binding the acceleration and the force density terms of a small fluid particle), where: **V** is the local velocity of translation (of the respective small particle) – the intensity of the local fluid field; $\mathbf{\Omega} = \nabla \times \mathbf{V} = 2\,\boldsymbol{\omega}$ is the vortex (curl **V**), with **ω** the local instantaneous velocity of rotation (of the above particle), a vector function which may be either continuous (vortex volumes or vortex sheets) or discrete (like for the "Kármán vortex street" model, with vortex filaments of intensity - circulation - given by Stokes's formula: $\Gamma = \int_l \mathbf{V}\cdot d\mathbf{R} = \int_\sigma \mathbf{\Omega}\cdot\mathbf{n}\, d\sigma$; here σ is the area of an arbitrary section of a vortex tube, section lying on the closed curve *l*; **n** is the normal versor; $\sigma \to 0$); **f** is the mass force density (a conservative one – a gradient): $\mathbf{f} = \nabla(-\,gz) = -\,\nabla(gz)$; g is the acceleration due to gravity; z is the geometrical height (height of the point considered above a reference horizontal plane xOy); p is the fluid static pressure; ρ is the fluid density; t is the time.

For this inviscid fluid flow the momentum equation yields the Crocco–Vászonyi form:

$$\frac{\partial \mathbf{V}}{\partial t} + \mathbf{\Omega}\times\mathbf{V} = T\nabla S - \nabla(i_0 + gz) \quad , \quad \text{with} \quad i_0 = i + \frac{1}{2}V^2 \quad \text{– the total (stagnation) specific enthalpy;}$$

here $i = U + p/\rho$ is the specific enthalpy; U is the specific internal energy; V is the modulus of **V** ; S is the specific entropy; T is the static temperature (absolute) of the fluid particle. For a perfect (an ideal) gas: $p = \mathcal{R}\rho T$, with $\mathcal{R}$ = const.

This fluid motion takes place in the envelope sheet of the successive local planes osculatory to a flow certain streamline. The general equation above is valid for the *non-isentropic (rotational) flow of a certain (non-barotropic) fluid*. For a steady motion and for gases respectively, we have $\partial\mathbf{V}/\partial t = 0$ and $\mathbf{f} = 0$, thus leaving

$$\nabla\left(\frac{1}{2}\mathbf{V}^2\right) + \mathbf{\Omega}\times\mathbf{V} = -\frac{1}{\rho}\nabla p \quad , \quad \text{or} \quad \nabla\left(\frac{1}{2}V^2\right) + T\nabla S = -\frac{1}{\rho}\nabla p \quad , \qquad (1)$$

*Tel.: +40 021 434 0083; fax: +40 021 434 0082
E-mail address: rselescu@aero.incas.ro; *URL:* http://www.incas.ro

Proceedings of the XXXIInd "Caius Iacob" National Conference on Fluid Mechanics and its Technical Applications
2009, October 16 – 17, Bucharest, Romania
Richard Şelescu – Intrinsic analytic study on an isoenergetic flow of a compressible fluid
Part 2: Extension to some cases in magneto-plasma dynamics

Intrinsic analytic study on an isoenergetic flow of a compressible fluid Part 2: Extension to some cases in magneto-plasma dynamics

Richard Şelescu*
Department of Aerodynamics, "Elie Carafoli" National Institute for Aerospace Research – INCAS, 061126, Bucharest, Romania

Abstract: This part studies and clarifies some local phenomena in magneto-fluid dynamics. The model from part 1 was extended to magneto-plasma dynamics (taking into account the flow vorticity and the Joule–Lenz heat), considering a non-isentropic flow of an inviscid electroconducting fluid in an external magnetic field. There are some space curves along which the motion equation admits a first integral, introducing a new physical quantity – Şelescu's **$** vector. For a fluid having an infinite electric conductivity, these curves are the flow isentropic lines, enabling the treatment of any 3-D flow as a "quasi-potential" 2-D one. The case of unsteady flow (and electric field and charge, and magnetic field) of an inviscid electroconducting liquid was studied, giving an exact first integral for the motion equation. The model was extended to viscous and visco-magnetic flows of conducting fluids, introducing the "zero-work" surfaces, two new physical quantities – Şelescu's **§** and **Ş$_m$** vectors, a new intrinsic coordinate system, and a 2-D magnetic "quasi-potential". A first integrability case in steady MHD (for both the motion and induction equations) was found. Almost all first integrals are similar to D. Bernoulli and D. Bernoulli–Lagrange ones. The PDEs of Şelescu's vector lines and zero-work surfaces are given.
2000 Mathematics Subject Classification: 70 Mechanics of particles and systems; 76 Fluid mechanics; 78 Optics, electromagnetic theory; 80 Classical thermodynamics, heat transfer; 85 Astronomy and astrophysics
Keywords: Rotational flows; Steady and unsteady flows; Inviscid and viscous fluids; Compressible fluids; Isentropic surfaces; Flow of an electroconducting fluid in an external magnetic field; Şelescu's vectors, space curves and zero-work surfaces

2.1. Introduction, nomenclature and the first approach to the new model in magneto-hydrodynamics

MHD models are extensively used in the analysis of magnetic fusion devices, industrial processing plasmas, and ionospheric/astrophysical plasmas. MHD is the extension of fluid dynamics to ionized gases, including the effects of electric and magnetic fields. So, the general model presented in the part 1 can be extended to some special (but usual) cases in magneto-plasma dynamics, considering an adiabatic but *non-isentropic* flow (taking into account the flow vorticity effects, as well as those of the Joule–Lenz heat losses) in an external magnetic field, obtaining *new first integrability cases* (similar to the D. Bernoulli ones): a) assuming a continuous medium, i.e. no (or at most binary elastic) particle collision (i.e. no chemical or nuclear reaction) is present, and also neglecting the energy variation (loss or increase) due to radiation; b) making no distinction between the intensity of the magnetic field and the magnetic induction of the medium (in the Gaussian system of units), since for all conducting fluids the magnetic permeability is approximately equal to 1 (see [12] – [14]); c) assuming that the value of the electric conductivity of the fluid medium is uniform and isotropic throughout and independent of the magnetic field intensity.

Analogously to the case for part 1, in the magneto-plasma dynamics (by plasma we understand a mixture of neutral and excited atoms, ions, electrons and photons), the general form of the differential equation of motion for (not as usual) *an adiabatic* but *non-isentropic flow of a barotropic inviscid electroconducting fluid* in an external electromagnetic field, *considering the flow vorticity*, is (see [12] – [14], [16], [17] for the right-hand side only)

$$\frac{\partial \mathbf{V}}{\partial t}+\nabla\left(\frac{1}{2}\mathbf{V}^2\right)+\mathbf{\Omega}\times\mathbf{V}=\mathbf{f}-\frac{\nabla p}{\rho}+\mathbf{f}_{eL} \quad , \quad \text{or} \quad \frac{\partial \mathbf{V}}{\partial t}+\nabla\left(\frac{1}{2}\mathbf{V}^2\right)+\mathbf{\Omega}\times\mathbf{V}=\mathbf{f}-\frac{\nabla p}{\rho}+\frac{\rho_e}{\rho}\mathbf{E}+\frac{1}{c\rho}\mathbf{j}\times\mathbf{H}$$

(for the left-hand side one usually writes $\partial\mathbf{V}/\partial t + (\mathbf{V}\nabla)\mathbf{V}$, except for [15] and [18]–unexploited), with:
∇ nabla (Hamilton's operator); t the time; and c the light speed in a vacuum, binding the acceleration and the force density terms of a fluid particle, where:

$\rho = n_a m_a + n_+ m_+ + n_- m_- = \rho_a + \rho_+ + \rho_-$ is the plasma density, with $m_a = m_+ + m_-$ the mass of a neutral atom;

*Tel.: +40 021 434 0083; fax: +40 021 434 0082
E-mail address: rselescu@aero.incas.ro; *URL:* http://www.incas.ro

Asymptotic analysis of a degenerate evolution system

Claudia Timofte*
Faculty of Physics, University of Bucharest, P.O. Box MG-11, Bucharest-Magurele, Romania

Abstract: The goal of this paper is to analyze the effective behavior of the solution of a nonlinear problem arising in the modelling of thermal diffusion in a two- component composite. We consider a periodic structure formed by two materials with different thermal properties. We assume that we have space- dependent nonlinear sources and that at the interface between our two materials the flux is continuous and depends in a dynamical nonlinear way on the jump of the temperature field. We prove that the asymptotic behavior of the solution of this problem, as the small parameter which characterizes the sizes of our two regions tends to zero, is governed by a degenerate system, with extra terms capturing the effect of the interfacial barrier, of the dynamical boundary condition and of the nonlinear sources.

2000 Mathematics Subject Classification: 35B27; 35K65.
Keywords: homogenization; composite materials; nonlinear sources; dynamical boundary conditions.

1. Introduction and setting of the problem

The aim of this paper is to analyze the effective behavior of the solution of a nonlinear problem arising in the modelling of thermal diffusion in a periodic structure formed by two materials with different thermal properties, separated by an active interface. We assume that we have nonlinear sources acting in one component and that at the interface between our two materials the flux is continuous and depends in a nonlinear way on the jump of the temperature field.

Let Ω be a bounded domain in $\mathbf{R}^n$ $(n \geq 3)$, with the boundary $\partial\Omega$ being a Lipschitz manifold, made by a finite number of connected components.

We consider that Ω is a periodic structure formed by two components, Ω^ε and Π^ε, representing two materials with different thermal features, separated by an interface S^ε. We assume that both Ω^ε and $\Pi^\varepsilon = \Omega \setminus \overline{\Omega^\varepsilon}$ are connected, but only Ω^ε reaches the external fixed boundary of the domain Ω. Here, ε represents a small parameter related to the characteristic size of our two regions.

Following H. I. Ene and D. Polisevski [9], let Y_1 be a Lipschitz open connected subset of the unit cube $Y = (0,1)^n$. Let $Y_2 = Y \setminus \overline{Y_1}$. We suppose that Y_2 has a locally Lipschitz boundary. Moreover, we assume that the intersections of the boundary of Y_2 with the boundary of the unit cell Y are identically reproduced on opposite faces of the cube, which are denoted, for any $1 \leq i \leq n$, by

$$\Sigma_i = \left\{ y \in \partial Y \,\middle|\, y_i = 1 \right\}$$

and

$$\Sigma_{-i} = \left\{ y \in \partial Y \,\middle|\, y_i = 0 \right\}.$$

We suppose that repeating Y by periodicity, the union of all the sets $\overline{Y_1}$ is connected and has a locally C^2 boundary. Also, we assume that the origin of the coordinate system is set in a ball contained in this union. Let

$$Z_\varepsilon = \left\{ k \in \mathbf{Z}^n \,\middle|\, \varepsilon k + \varepsilon Y \subseteq \Omega \right\}$$

and

$$I_\varepsilon = \left\{ k \in \mathbf{Z}_\varepsilon \,\middle|\, \varepsilon k \pm \varepsilon e_i + \varepsilon Y \subseteq \Omega,\ \forall i = \overline{1,n} \right\},$$

where e_i are the elements of the canonical basis of $\mathbf{R}^n$.

* Tel. : +4021 457 4949; fax: +4021 457 4521
E-mail address: claudiatimofte@yahoo.com

Index

A

ALEXANDRESCU Maria, 1
ALEXANDRESCU Nicuşor, 1
ANDREI Irina Carmen, 16, 206
ANDREI Marina, 159
ANDREIAN Bogdan, 63
ANDREIAN Dan Andrei, 63
APOSTOLESCU Nicolae, 29

B

BARAN Daniela, 29
BARBELIAN Mihai, 16
BERBENTE Corneliu, 38
BERBENTE Sorin, 38
BRÎNZĂ Ionuț, 138
BOBONEA Andreea, 46
BUTOESCU Valentin, 55

C

CARABINEANU Adrian, iii, iv
CARDOŞ Vladimir, 109
CAZACU Dimitrie Mircea, 63
CÂRLĂNESCU Cristian, 73
CERCEL Cristian, 159
CHIRILĂ I. Aurel, 83
CRUCEANU Ştefan-Gicu, 90

D

DOBRE Alin, 148
DUMITRACHE Alexandru, 102, 109, 193
DUMITRESCU Horia, 102, 109, 193

F

FRUNZULICĂ Florin, 102, 109

G

GRĂDINARU Stelian, 115

H

HRIȚCU Eusebiu Constantin, 73

I

IONESCU Mircea Dan, 159
IORGA Ionel Cristinel, 125

J

JUNCU Gheorghe, 132

L

LĂNCRĂNJAN Ion, 138

M

MĂRIEŞ Ecaterina, 179
MOCANU Daniel, 148
MOREGA M. Alexandru, 83, 148, 172
MOREGA Mihaela, 148

N

NAE Cătălin, 152
NICOLA Aurelian, 132
NICULESCU Mihai Leonida, 159

P

PETRE V. Marin, 172
POPA Constantin, 132
POPESCU Alin Gabriel, 179
POPESCU Dumitru, 179
PORUMBEL Ionuț, 73
PREOTU Octavian, 193

S

STANCIU Virgil, 206
STATE Claudia Alice, 216
STROE Gabriela Liliana, 16

Ş

ŞELESCU Richard, 224
ŞELESCU Richard, 239

T

TIMOFTE Claudia, 259

ISSN 2067 – 4414
ISSN National Center
Romanian National Library

ORGANIZING COMMITTEE

Prof. Dr. Corneliu Bălan

E-mail: balan@hydrop.pub.ro

"Politehnica" University of Bucharest

http://www.pub.ro

Prof. Dr. Adrian Carabineanu

E-mail: acara@fmi.unibuc.ro

University of Bucharest

http://www.unibuc.ro/

Dr. Stelian Ion

E-mail: ro_diff@yahoo.com

"Gheorghe Mihoc - Caius Iacob" Institute of Mathematical Statistics and Applied Mathematics of Romanian Academy

http://www.ima.ro/

Dr. Richard Şelescu

E-mail: rselescu@incas.ro

"Elie Carafoli" National Institute for Aerospace Research – INCAS

(under the Aegis of Romanian Academy)

http://www.incas.ro

CONTENTS

- Maria Alexandrescu, Nicuşor Alexandrescu – Ice accretion simulation on NACA-0012 ……… 1
- Irina Carmen Andrei, Mihai Barbelian, Gabriela Liliana Stroe – Jet engines thrust control system analysis ……… 16
- Daniela Baran, Nicolae Apostolescu – Numerical aspects in dynamic response of an elastic plane ……… 29
- Corneliu Berbente, Sorin Berbente - An extension of Hugoniot – Rankine relations for reacting and nonreacting flows considering the variation of specific heats with temperature ……… 38
- Andreea Bobonea – Computational fluid dynamics analysis of flow over a re-entry blunt body ……… 46
- Valentin Butoescu – An elastic model of a textile surface ……… 55
- Mircea Dimitrie Cazacu, Dan Andrei Andreian, Bogdan Andreian – Aquaplaning of airplane tyre ……… 63
- Cristian Cârlănescu, Ionuţ Porumbel, Constantin Eusebiu Hriţcu – Linear eddy model numerical simulation of the turbulent flow in a round turbulent jet ……… 73
- Aurel I. Chirilă, Alexandru M. Morega – Electric motor frame cooling; a finite element analysis ……… 83
- Stefan-Gicu Cruceanu – Arclength continuation for implicitly defined curves ……… 90
- Alexandru Dumitrache, Florin Frunzulică, Horia Dumitrescu – A Numerical analysis of combustion process in an axisymmetric chamber ……… 102
- Florin Frunzulică, Alexandru Dumitrache, Horia Dumitrescu, Vladimir Cardoş – Unsteady free-wake vortex particle model for horizontal axis wind turbines ……… 109
- Stelian Grădinaru – Self-propulsion of thin profiles; an analytic approach ……… 115
- Ionel Cristinel Iorga – The effect of saturation on the longitudinal command in the approaching phase of flight for landing ……… 125
- Gheorghe Juncu, Aurelian Nicola and Constantin Popa – A numerical study of laminar flow past two circulating spheres in tandem 132
- Ion Lăncrănjan, Ionuţ Brînză – Applications of visualization methods in aeronautics and civil engineering ……… 138
- Alexandru M. Morega, Alin Dobre, Mihaela Morega, Daniel Mocanu – Finite element modeling of blood flow in an arterial bifurcation ……… 148
- Cătălin Nae – Noise metric for high-lift-low-noise concept validation ……… 152
- Mihai Leonida Niculescu, Mircea Dan Ionescu, Marina Andrei, Cristian Cercel – Analysis of aerodynamic field and noise from a ducted wind turbine ……… 159
- Marin V. Petre, Alexandru M. Morega – A numerical study of the heat transfer by the aluminum electrolysis cell shell ……… 172
- Dumitru Popescu, Alin Gabriel Popescu, Ecaterina Mărieş – Mathematical modelling of a giant unilamellar vesicle under positive osmotic pressure ……… 179
- Octavian Preotu, Alexandru Dumitrache, Horia Dumitrescu – Similar solution for a Coanda flow and applications to directed synthetic jets ……… 193
- Virgil Stanciu, Irina Carmen Andrei – Optimization and stability issues in simulating the running regimes of the jet engines ……… 206
- Claudia Alice State – The influence of saturation relative to the aileron control in the roll motion ……… 216
- Richard Şelescu – Intrinsic analytic study on an isoenergetic flow of a compressible fluid
  Part 1: "Isentropic" and "zero work" surfaces in aero-gas dynamics ……… 224
- Richard Şelescu – Intrinsic analytic study on an isoenergetic flow of a compressible fluid
  Part 2: Extension to some cases in magneto-plasma dynamics ……… 239
- Claudia Timofte – Asymptotic analysis of a degenerate evolution system ……… 259
- Index ……… 265

ISSN 2067 – 4414

ISSN National Center

Romanian National Library

BUCHAREST

2009