INCAS - National Institute for Aerospace Research "Elie Carafoli"
(under the Aegis of the Romanian Academy)

Proceedings
of
the XXXIIIrd "Caius Iacob" National Conference
on
Fluid Mechanics and its Technical Applications

2011, September 29 – 30

Bucharest, Romania

Organizers

INCAS - National Institute for Aerospace Research "Elie Carafoli" (under the Aegis of the Romanian Academy)

University of Bucharest

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

"Politehnica" University of Bucharest

BUCHAREST

2011

Proceedings of
the XXXIIIrd “Caius Iacob” National Conference on FLUID MECHANICS and its TECHNICAL APPLICATIONS

Bucharest, Romania, September 29 – 30, 2011

Sessions:

Mathematical Modeling & Numerical Simulations;
Dynamical Systems; Technical Applications

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

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

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

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

2011, September 29 – 30

Bucharest, Romania

Organizers

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

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

University of Bucharest

“Politehnica” University of Bucharest

BUCHAREST

2011

CONTENTS

CONTENTS

An Approximate Integral Scheme of Calculating the Transitional Boundary Layer in Two-Dimensional Incompressible Flow

Sakhr Abu-Darag* and Vladimír Horák**
Department of Mechanical Engineering, University of Defence in Brno, Kounicova 65, 662 10 Brno, Czech Republic

Abstract: Integral calculations of two-dimensional, incompressible, thermal, transitional boundary layers have been performed. To precede these approximate calculations, mathematical model was developed in order to enable prediction of the main boundary layer integral parameters. The model was proposed to calculate the characteristics of the boundary layers under the effect of local heat transfer and moderate free-stream turbulence levels by enhancing established integral techniques in conjunction with intermittency weighted model of the transitional boundary layer. Empirical relationships for the prediction of the start and end of transition, as well as the development of the boundary layer during the transition region were based on results of experimental investigations. Since the heat transfer coefficient between external flow and surface is extremely influenced by the level of turbulence in the flow, it is also found to be very sensitive to the solid surface temperature and thereby an adequate solution of the thermal boundary layer is required. To satisfy these conditions, the mathematical model included both dynamic and thermal boundary layer equations in integral form. To support the results validation, a numerical investigation utilized Menter el. al [7] model in ANSYS-CFX tool has been represented beside the experimental results.

2010 Mathematics Subject Classification: 31 Potential theory; 45 Integral equations; 76 Fluid mechanics; 80 Classical thermodynamics, heat transfer
Keywords: Transition onset; Intermittency; Integral scheme; Transport equation model; Boundary layer; Thermal boundary layer; Turbulence intensity

1. Introduction

The subject of laminar-turbulent transition is of considerable practical interest and has a wide range of engineering applications due to the fact that transition controls the evolution of important aerodynamic quantities such as drag or heat transfer. Transition in boundary layer flows in turbomachines and aerospace devices is known to be affected by various parameters, such as freestream turbulence, pressure gradient and separation, Reynolds number, Mach number, turbulent length scale, wall roughness, streamline curvature and heat transfer. Due to this variety of parameters, there is no mathematical model exist that can predict the onset and length of the transition region. In addition to the influence of these parameters upon transition origination, the poor understanding of the fundamental mechanisms which lead initially small disturbances to transition may also caused this lack. At present, there are three main concepts used to model transition in industry. The first approach is based on the stability theory where the successful technique is so-called e^N method. This method is based on the local linear stability theory and the parallel flow assumption in order to calculate the growth of the disturbance amplitude from the boundary layer neutral point to the transition location. A shortcoming of this technique indicates that it is not compatible with the current CFD methods because the typical industrial Navier-Stokes solutions are not accurate enough to evaluate the stability equation. Moreover since it is based on the linear stability theory, it cannot predict the transition due to non-linear effects such as high freestream turbulence or surface roughness. The second approach uses the conventional turbulence models such as the two-equation turbulence model of Launder and Sharma [1]. The disadvantages of this solution that first, ignores the transition physics and the importance of the transition zone completely and secondly, it is fabricated especially to deals with flows where the transitional region covers a large portion of the flow field. The main concept of the construction of these models that, the calibration of the damping functions in these models is based on reproducing the viscous sublayer behavior, not on predicting transition from laminar to turbulent flow. The last

*Tel.: +420 973 442 410; fax: +420 973 442 613
E-mail address: sakhar.darag@unob.cz *; URL:* http://www.unob.cz
**Tel.: +420 973 442 616 ; fax: +420 973 442 613
E-mail address: vladimir.horak@unob.cz*; URL:* http://www.unob.cz

Lift and Thrust Generation on a Cycloidal "Flapping" Wing

Valentin Butoescu*

INCAS- National Institute for Aerospace Research "Elie Carafoli", B-dul Iuliu Maniu 220, Bucharest 061126, Romania

Abstract: A quasi-steady model for the flow analysis has been used in order to find the main characteristics of a special "flapping" mechanism proposed by the author.

2010 Mathematics Subject Classification: 31 Potential theory; 70 Mechanics of particles and systems; 76 Fluid mechanics

Keywords: aerodynamic forces; unsteady lift generation; cycloidal "flapping" wing; flow analysis; flapping mechanism; quasi-steady model

Introduction

In a previous paper [1] we presented a new device that generates both lift and thrust for **m**icro **a**ir **v**ehicles (or MAV's).

This mechanism was called "modified cycloidal wing device"** or a cycloidal rotor and is based on the *cycloidal propulsor*, invented by Schneider in 1926, [2].

Although the device uses a rotary wing system it is not a helicopter type of MAV, firstly because the rotation axis is not vertical but horizontal.

The author considers it closer to a modified flapping system, when the wing tip does not describe an elongated curve but a circle. That is why we put the word *flapping* inside the quotation marks.

The present paper contains a method for calculating the aerodynamic forces on the wings based on the simplified assumptions of the quasi-steady model.

A new wing mechanism for unsteady lift generation

It is known that the insect/ hummingbird wing tip describes a closed curve that is very elongated and has different shapes.

Consider now a mechanism that makes the wing tip to move along a general closed curve, not necessarily an elongated one.

During this motion of "flapping" which has transformed now in a rotary motion, the angle of attack of the wing changes continuously so that a purpose is achieved. This purpose is to obtain a mean force that equals both the weight and the drag of the MAV.

For this kind of mechanism, the wing radius remains always on a surface like a conical frustum.

The "flapping" motion presented before can be thought of as a limiting case when the cone degenerates to a plane (stroke plane).

A flapping device is made up of two mechanisms: one that generates the flapping motion and the second that rotates the wing about its longitudinal axis (pitching motion).

If we use an electric motor, we have to transform a rotation into a reciprocating motion and doing so, to get the "flapping" motion.

Besides we need another mechanism to generate the wing rotation (or pitching). The goal of the present approach is to avoid the first mechanism.

So we searched for a simpler mechanism that uses directly the circular motion rather than a reciprocating motion.

There is such a mechanism. This mechanism has been invented by the Austrian engineer Schneider in 1926, [2]. It is used for propulsion and maneuvering system of some ships.

In the following considerations we adopted the simplified assumptions of the quasi-steady model.

Therefore the instantaneous aerodynamic forces are regarded as equal to the forces occurring in the steady motion at the same angles of attack.

* Tel.: +40 21 434 0083; fax: +40 21 434 0082
E-mail address: vbutoesc@incas.ro; *URL:* http://www.incas.ro

** The author has applied for a patent in Romania which will cover both the mechanism and the wing setting.

The screw axial force maximization reported to its mechanical driving power for a familiar airplane

Mircea Dimitrie Cazacu*
"POLITEHNICA" University of Bucharest, Splaiul Independentei 313, 060042, Bucharest, Romania

Abstract: Due to the important role of the airplane screw efficiency, concerning the energy and fuel economy, the airplane action range increasing and the environment pollution diminishing [1], [2], I shall present an original method to maximize the screw performances, establishing even the optimum blade profile, its relative β optimum angle and its setting optimum angles *i* for different blade radius, by annulment of the partial differential of the same report between the axial force and mechanical power consumption, but now concerning the attack angle *i*. Finally one gives the best characteristics for many aero dynamical profiles.

2010 Mathematics Subject Classification: 35 Partial differential equations; 76 Fluid mechanics
Keywords: axial force; mechanical power; airplane; lift; drag; velocity; aerodynamic resultant force; fineness of the profile

1. The classical expressions of the lift and drag components of the aerodynamic resultant

Taking into account the expressions of the lift and drag components of the aerodynamic resultant force over an airplane screw profile (fig. 1),

$$F_y = c_y(i)\frac{\rho}{2}W^2 b\,l(R), \qquad F_x = c_x(i)\frac{\rho}{2}W^2 b\,l(R), \tag{1}$$

exerted on the profiled blade, laid at the incidence angle *i* with respect to the relative angle β, corresponding to the relative velocity *W* from the velocity triangle,

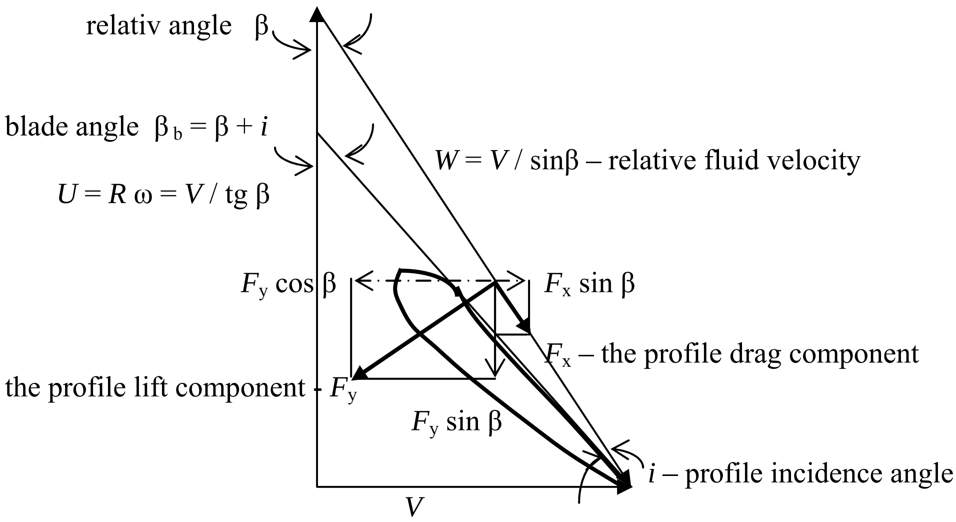

Fig. 1 The velocity triangle and the hydrodynamic resultant components

one can calculate the axial component of these forces, representing the propulsion force

$$F_a = F_y\cos\beta - F_x\sin\beta = \frac{\rho}{2}V^2 bl\left[c_y(i)\frac{\cos\beta}{\sin^2\beta} - c_x(i)\frac{1}{\sin\beta}\right], \quad f_a = \frac{2F_a}{\rho V^2 bl} = c_y(i)\frac{\cos\beta}{\sin^2\beta} - c_x(i)\frac{1}{\sin\beta} \tag{2}$$

and also the expression of the negative shaft driving mechanical power

$$P_m = U(F_y\sin\beta + F_x\cos\beta) \quad \text{or} \quad p_m = \frac{2P_m}{\rho V^3 bl} = c_y(i)\frac{\cos\beta}{\sin^2\beta} + c_x(i)\frac{\cos^2\beta}{\sin^3\beta}. \tag{3}$$

* *E-mail address:* cazacu.dimitrie@yahoo.com

Analysis of the swirling flow at GAMM Francis runner outlet for different values of the discharge

Tiberiu Ciocan*, Romeo F. Susan-Resiga**, Sebastian Muntean***
*Department of Hydraulic Machinery, Politehnica University of Timisoara Bd. Mihai Viteazu 1, 300222, Timisoara, Romania
**Department of Hydraulic Machinery, Politehnica University of Timisoara Bd. Mihai Viteazu 1, 300222, Timisoara, Romania
***Centre for Advanced Research in Engineering Sciences, Romanian Academy – Timisoara Branch Bd. Mihai Viteazu 24, 300223, Timisoara, Romania

Abstract: Nowadays an important, and still open, issue in Francis runner design is the blade geometry at the trailing edge, wich directly determines the swirling flow at the runner outlet and further downstream the pressure recovery and hydraulic losses in the turbine draft tube. In this paper we present and validate a novel mathematical model for computing the radial profiles of both axial and circumferential velocity components, respectively, at the runner outlet for Francis hydraulic turbines within the full operating range. We apply this methodology of computing the swirling flow downstream the runner, over the GAMM Francis turbine model, where experimental data were available. We present the computation procedure for the dimensionless discharge and dimensionless flux of moment of momentum, respectively. We introduce the swirl-free velocity profile in order to expres the relationship between the axial and circumferential velocity components. This hypothetical velocity, the swirl-free velocity, is subject of optimization in order to achieve the best performance of the draft tube over the whole intended operating range.
2000 Mathematics Subject Classification: 45 Integral equations; 65 Numerical analysis; 76 Fluid mechanics
Keywords: swirling flow; Francis turbine; flux of moment of momentum; swirl-free velocity

1. Introduction

Nowadays, modern hydraulic turbines meet new challenges associated with variable demand on the energy market and limited possibilities for energy storage, resulting in the need for flexibility in their operation. This need for flexibility in the operation of a turbine, means that there is often a tendency (required) to use the turbines for a wide range of operating regimes, far from the operating point for which have been designed. Hydraulic turbines, especially radial-axial turbines (e.g. GAMM Francis turbine [1]) having a rotor with fixed pitch blades, have a sharp drop in efficiency and severe pressure fluctuations at off-design operating regimes. When operating at part-load, in the turbine draft tube, it appears a high-level of residual swirl, as a result between the mismatch of the swirl configuration at the runner inlet imposed by the guide vanes, and the angular moment extracted by the turbine runner [2].

The helical vortex (which itself is part of the flow rotation) appears due to the decelerated flow rotation and it is the source of flow unsteadiness with associated pressure fluctuations, that can lead in breakage of the runner blades. Evaluation of the turbine efficiency for the full turbine operating range, depending on the discharge coefficient and energy coefficient with respect to the IEC regulations [3], is still the standard experimental investigation on model turbines in order to predict the performance of the real size machine, and the resulting efficiency hill-chart usually displays a peek efficiency, wich is called the “best efficiency point” (BEP).

Due to technological advances in recent decades, numerical simulation can be made on all components of a Francis turbine, which allow a high accuracy analysis of the spiral, the guide vanes, the rotor and draft tube, respectively [4].

For this reason in the last 20 years, it appeared numerous programs dedicated to investigating the hydraulic losses wich occurs in the draft tube, such as:

- GAMM Workshop – „3D - Computation of Incompressible Internal Flows” [5]
- Turbine – 99 Workshop on draft tube [7]
- FLINDT Project – Flow investigation in draft tube [8]

Better understanding of the particularities of the swirling flow at the runner outlet / draft tube inlet, is the key in developing new techniques for controlling the swirling flow downstream the Francis turbine runner.

* Tel.: +40 256 403692; fax: +40 256 403692
E-mail address: ciocan@mh.mec.upt.ro; *URL:* http://mh.mec.upt.ro/cnisfc/

Rheology of yield stress fluids: experiments, constitutive relation, numerical simulations

Daniela Coblas*, Diana Broboană** and Corneliu Bălan***
Politehnica University, DBB – REOROM Laboratory, Splaiul Independentei 313, 060042 Bucharest, Romania

Abstract: The non-monotonic character of the steady flow curve is a possible explanation for spurious phenomena (wall depletion, **yield stress**, stick-slip flow and shear banding) observed in the shear rheological tests of complex fluids with unstable internal network structure. In this work, lubricating greases, polyxiloxane and cosmetic creams, sensitive magnetic fluids and solder pastes are experimentally investigated to detect possible existence of instability region in the flow curve. The experiments are based on simple shear and dynamic tests performed both in the linear and non-linear flow regimes. The results showed the time dependence and the existence of some discrete steady preferred values of the viscosity function. The experiments are accomplished by numerical simulations of generalized Newtonian models with non-monotonous flow curve. The correlation between experiments and simulations indicate that material instability is always related with the onset of banding flow structure in the vicinity of the walls. The numerical results are qualitatively consistent with experiments and prove that shear bands location within the gap is dependent (for given material parameters) by the dynamics of the flow, respectively the values of initial/ boundary conditions.

2010 Mathematics Subject Classification: 35 Partial differential equations; 76 Fluid mechanics; 78 Optics, electromagnetic theory
Keywords: yield stress fluids; numerical simulations; sensitive magnetic fluids; flow regimes; experiments; dynamics of complex fluids; electro- and magneto-rheological fluids; Newtonian viscosity region

1. Introduction

The present paper investigated the dynamics of complex fluids in simple shear tests. The materials under investigations (lubricating greases, creams, sensitive magnetic fluids and solder pastes) are basically dispersions, characterized by internal deformable network structures, common included in the class of *soft solids*. These particular samples start to flow at certain levels of shear stress and strain, therefore they are sometimes called fluids with *yield stress* - σ_0, [1], [2].

It is important to mention that almost all electro- and magneto-rheological fluids disclose a yield stress, beyond a critical level of the applied field, [3].

Fluids with yield stress have a long history in rheometry and still is a debate on their real existence, [1], [4]. Nevertheless, the dynamics of that materials evidence in simple shear geometry spurt phenomena as wall depletion, stick-slip flow and shear banding.

During the tests is always observed a yielding state and a particular value of the shear stress which separates the high Newtonian viscosity region ($\sigma < \sigma_0$ from the pronounced shear thinning domain ($\sigma > \sigma_0$), see Fig. 1.

The threshold value σ_0 determine a jump in strain rate value (see the marked points in Fig. 1) and the onset of the material flow behavior.

This rheology is common associated to a non-monotonous flow curve generated by a material unstable constitutive relations, [1].

The most common model from this category is the Johnson and Segalman differential relation, see [5], [6].

The paper is dedicated to the measurement and modeling of materials characterized by non- monotonous flow curve.

The goal of the study is to test the capabilities of the generalized Newtonian model to represent phenomena as shear banding formation in the gap of rheometer.

* *E-mail address*: daniela.coblas@yahoo.com,
** *E-mail address*: diana.broboana@upb.ro,
*** *E-mail address*: corneliu.balan@upb.ro

Wave-wave regular interactions of a gasdynamic type

Liviu Florin Dinu[*], Marina Ileana Dinu[**]
[*]*Institute of Mathematics of the Romanian Academy, 21, Calea Grivitei Street, Bucharest, Romania*
[**]*Politechnical University of Bucharest, 313, Splaiul Independentei Street, Bucharest, Romania*

Abstract: Two gasdynamic analytic approaches [of a Burnat type / Martin type] are respectively used in order to construct two analogous and significant pairs of classes of solutions [isentropic pair / anisentropic (of a particular type) pair]. Each mentioned pair puts together a class of "wave" elements and a class of "wave-wave regular interaction" elements. A classifying parallel is finally constructed between the two analogous pairs of classes -- making evidence of some consonances and, concurrently, of some nontrivial contrasts.
2010 Mathematics Subject Classification: 35A30, 35L60, 35L65, 35Q35, 76N10, 76N15.
Keywords: geometrical approach; two-dimensional solutions; quantifiable "amount" of genuine nonlinearity; regular interaction vs. irregular interaction: euristic details.

1. Introduction

Finding a solution to a quasilinear system of a gasdynamic type

$$\sum_{j=1}^{n}\sum_{k=0}^{m} a_{ijk}(u)\frac{\partial u_j}{\partial x_k} = 0, \qquad 1 \le i \le n \tag{1}$$

[ex. Euler isentropic /anisentropic; possibly multidimensional] is a hard task generally. Our ***talk*** has considered constructively, in presence of certain integrability restrictions, some highly nontrivial and *significant* classes of solutions to such type of systems.

Two analytic approaches have been considered in our talk: a Burnat type "algebraic" and genuinely nonlinear approach [structured by a duality connection between the hodograph character and the physical character] and a Martin type two-dimensional "differential" approach [associated with a Monge−Ampère type representation].

A pair of significant classes of solutions have been associated in our talk to *each* of the two mentioned approaches.

In the *isentropic* case a Burnat type approach has been used to constructively structure: ● some *simple waves solutions* − here called *waves* [a first significant class], ● some *wave-wave regular interaction solutions* [a second significant class]; and, ● a *multidimensional extension* of the two classes mentioned above − with a *classifying potential*; a *regular* character of the wave-wave interaction described appeared to essentially reflect facts of a multidimensional and skew construction.

In the *anisentropic* case − and in two independent variables − a Martin type approach has been used, as associated with a *particular* gasdynamic example, to constructively structure an *anisentropic* **analogue** of the isentropic pair of classes mentioned above: the *anisentropic pair* which puts together ● some *pseudo simple waves solutions* [a first significant class] and ● some *pseudo wave-wave regular interaction solutions* [a second significant class]. Details concerning the nature of the mentioned analogous character have been presented.

A *classifying parallel* has been presented concurrently in our talk between the two analogous pairs of classes [isentropic, anisentropic] − making evidence of some *consonances* and, respectively, of some *nontrivial contrasts* of the two mentioned constructions [Burnat type, Martin type].

The regular passage [which uses the two analogous pairs of classes] from an isentropic description to an anisentropic description appeared to be **fragile**. Our talk also presented some essential details of this fragility.

The present ***Proceedings paper*** [a small fragment of our talk] includes two selfsimilar isentropic exemples − significant and highly nontrivial − of wave-wave regular interaction solutions. We use these examples to suggest that a certain structure could be associated with each "amount" of genuine nonlinearity eventually available and that there is a *hierarch*y of such structures.

The two types of wave-wave regular interactions constructed in our talk [isentropic /anisentropic] appeared to parallel, from an *analytic, local* and *regular* prospect, some details [interactions of simple waves solutions] of the Zhang and Zheng two-dimensional *qualitative, global* and *irregular* construction. The two examples in the

[*] *E-mail address*: liviu.dinu@imar.ro; *URL*: http://www.imar.ro/~lfdinu/
[**] *E-mail address*: marinadinu@gmail.com;

Blood Flow – Vessel Interaction in a Subclavian Aneurysm

Alin A. Dobre* and Alexandru M. Morega**

*Faculty of Electrical Engineering, University POLITEHNICA of Bucharest, Romania, alin.dobre@iem.pub.ro.
**Faculty of Electrical Engineering, University POLITEHNICA of Bucharest; "Gheorghe Mihoc – Caius Iacob" Institute of Statistical Mathematics and Applied Mathematics, Bucharest, Romanian Academy, Romania, amm@iem.pub.ro.

Abstract: This paper presents a mathematical mode and numerical simulation results of the blood flow-structural interaction, which occurs in a saccular aneurysm emerging out of the left subclavian artery, using computational domains made of by medical images reconstruction. A correlation between the total force per area acting upon the artery walls by the pulsatile blood flow and the rupture probability are also investigated.

2010 Mathematics Subject Classification: 76 Fluid mechanics; 92 Biology and other natural sciences
Keywords: Saccular aneurysm; blood flow; numerical simulation; finite element; CT image reconstruction

1. Introduction

Aneurysms are balloon-like bulges that occur in different types of arteries due to the thickening of the blood vessels' walls. The causes of these pathological formations are not yet well understood: some aneurysms are congenital while others occur there where the arteries walls withstand higher blood pressure, higher levels of cholesterol or atherosclerotic disease [1-3].

The blood vessel walls have a certain thickness adapted to body region they cross, in order to withstand the normal blood pressure.

The artery walls may be damaged or injured due to genetic conditions or trauma, giving birth to regions prone to aneurysm formation.

Also, high blood pressure, high levels of cholesterol and atherosclerotic disease are important factors, which influence the aneurysm formation, may grow larger and rupture or dissect.

A ruptured aneurysm may cause massive bleeding inside the body, while the aneurysm dissection manifests as a split in one or more layers of the artery walls causing bleeding into and along the layers of the arteries walls. Usually, both rupture and dissection are fatal.

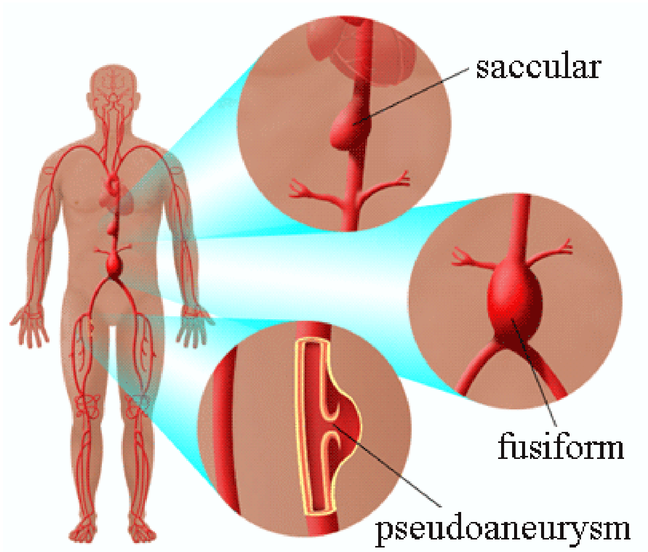

Fig. 1 Different types of aneurysms and the body regions prone to their formation, from [4]

Aneurysms could develop in any region of the body. Yet, there are two most common types of aneurysms: intracranial or cerebral aneurysms, which occur inside the brain, and aortic aneurysms, which develop inside the aorta [5].

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

The development of an aerodynamically efficient chevron for super circulation turbofan wing – the rhino chevron

Valeriu Drăgan*
Department of Aerospace Sciences - Faculty of Aerospace Engineering - "Politehnica" University of Bucharest, Romania

Abstract: Current non-regional airliners rely on turbofan technology for propulsion. The main concerns regarding noise pollution with regard to these engines is the fan flow interaction noise.
The phenomenon that causes this noise has been analyzed and modeled mathematically since Sir Cyril Lighthill's eight power law. Mitigation of the interaction noise has numerous technical solutions, of which chevron technology stands out. However, airlines are hesitant to adopt these solutions because of their poor aerodynamic performance. For this reason the last few years have seen the development of retractable chevron technologies, such as mechanical retraction and shape memory alloy chevrons. Such technologies are far from solving the efficiency issue, because both require constant maintenance.
This paper tries a different approach: by studying the way chevrons interact with the flow field, we could determine the mechanisms that lead to aerodynamic losses and try to eliminate them.
Aerodynamic losses have been found in regular chevrons to be caused by a pair of parasitic vortices on the top side of the chevron. By placing a divider between them we were able to eliminate them almost completely. Preliminary results are encouraging for this low cost high aerodynamically efficient chevron.
2000 Mathematics Subject Classification: 70 Mechanics of particles and systems, 76 Fluid mechanics
Key words: aero-acoustics, chevrons, circulation control, RANS, turbofan engine

1. Introduction

This paper tries to investigate ways to reduce turbofan noise by understanding the aerodynamics of turbofan noise generation and also seek to provide with a low cost solution for mitigating the generating mechanisms.

The main noise output of the turbofan is due to the Kelvin–Helmholtz phenomenon which occurs whenever two fluids in contact have different velocities.

This phenomenon was first described in a mathematical way by Sir Michael James Lighthill who gave the so-called "eighth power law" Eq. (1).

Since then, various refinements have been made to insure a closer link to the natural phenomenon. In this paper we will use Proudman Eq. (2) and Lilley's Eq. (3) for calculating both the Sound Pressure Level (SPL) and Sound Intensity Levels (SIL). Both equations were tried initially, however the Proudman equation turned out to be in better agreement with the known SPL of commercial high by-pass turbofan aircraft.

In order to mitigate jet interaction noise a series of proposals have been made including chevrons and fluid chevrons. This paper will investigate how chevrons interact with the flow from the turbofan and also compare conventional chevrons to an original new chevron, optimized for reducing drag and parasitic turbulence: the rhino chevron.

As it will become apparent, virtually any chevron – and in fact all noise reduction devices – will induce a different, lower noise as it mitigates the main one.

Using the two above mentioned formalisms we will also compare the noise made by conventional chevron interaction and the proposed rhino chevrons.

$$I(x) = \rho_0 \frac{V^8}{c^5}\left(\frac{L}{x}\right)^2 \tag{1}$$

where: $I(x)$ is the sound intensity observed at the distance x from the source;
V is the flow velocity difference;
c is the velocity of sound in the flow;
L is the diameter of the nozzle;

$$P = \alpha\rho\left(\frac{V}{c}\right)^5 \cdot \varepsilon \tag{2}$$

* *E-mail address*: drvaleriu@gmail.com

New model for inboard stall-delay

Horia Dumitrescu* and Vladimir Cardoş
Department of Fluid Mechanics, "Gh. Mihoc-C. Iacob" Institute of Statistical Mathematics and Applied Mathematics of the Romanian Academy, 050711, Bucharest, Romania

Abstract: Stall occurs at an airfoil when flow separates at high angles of attack, typically greater then 15^0; beyond this angle the 2-D lift coefficient drops significantly. There are significant differences between 2-D airfoil flow and 3-D flow on a rotating blade particularly for the innermost part of the blade. Lift coefficients attained at the inboard sections of a rotating blade are in excess of the maximum value possible in 2-D static test for post-stall angles of attack. This behavior is referred to as the stall delay phenomenon. The lack of a conceptual model for the complex 3-D flow field on the rotor blade, where stall begins, and how it progresses, has hindered the finding of a unanimously accepted solution. This work aims at giving a better understanding of the delayed stall events.

2000 Mathematics Subject Classification: 76 Fluid mechanics
Keywords: Three-dimensional boundary layer; Rotor aerodynamics; Stall-delay; Rotational effects; Wind turbines

1. Introduction

During the design process of a wind turbine blade, accurate and reliable prediction methods are required for the machine's full range of operating conditions. As engineering methods, blade element momentum (BEM) and vortex lattice (VL) methodologies have been widely used for rotor design and analysis. However, these methods are both based on the wind tunnel tested two-dimensional (2-D) aerodynamic coefficients, which do not include the three-dimensional (3-D) rotational effects. There are significant differences between 2-D airfoil flow and 3-D flow on a rotating blade particularly for the innermost part of the blade. Lift coefficients attained at the inboard sections of a rotating blade are significantly in excess of the maximum value possible (stall) in 2-D static test. That is to say, the angle of attack at which stall occurs is greater for a rotating blade. This behavior is referred to as the stall delay phenomenon which vanes along the blade with the augmented lift decreasing from blade root to tip. In order to obtain accurate rotor sectional lift characteristics and hence accurate power prediction, the 2-D airfoil data needs to be corrected for the highly 3-D inboard stall delay effects.

Since this stall delay phenomenon was first observed by Himmelskamp on propeller blades [1]. Researchers in the helicopter and wind turbine fields have paid close attention to it and have developed some empirical stall-delay models to better predict rotor performance. The stall delay phenomenon is, however, still far from being completely understood

2. Models for delayed stall regime

Stall occurs at an airfoil when flow separates at high angles of attack, typically >15^o, beyond this angle the 2-D lift coefficient drops significantly. It has long been known [1], [2], [3], however, that the observed power curve under conditions where most parts of the blade are stalled is consistent with significantly higher lift coefficients. The lack of a conceptual model for the complex 3-D flow field on the rotor blade, where stall begins, and how it progresses, has hindered the finding of a unanimously accepted solution. This section aims at giving a better understanding of the delayed stall events.

Six different models [4] to correct the airfoil characteristics for stall delay, including a wide range of different assumptions, are described. The first three models discussed, i.e. those of Snel et al [2], Chaviaropoulos and Hansen [5] and Raj [6], correct the lift and drag coefficients *CL* and *CD* for 3-D effects in the following way:

$$\begin{aligned} C_{L,3-D} &= C_{L,2-D} + l\left(\Delta C_L\right) \\ C_{D,3-D} &= C_{D,2-D} + d\left(\Delta C_D\right) \end{aligned} \tag{1}$$

where l and d are functions of the differences ΔCL and ΔCD between the *CL,INV* obtained in the ideal fluid flow

* Tel.: +40 21 318 2433; fax: +40 21 318 2439
E-mail address: horiadumitrescu@yahoo.com; *URL:* http://www.ima.ro

Smart concepts for actuation system and its control in a morphing wing

Teodor Lucian Grigorie, Andrei Vladimir Popov and Ruxandra Mihaela Botez[1]*
École de Technologie Superieure, Montreal, Quebec H3c 1k3, Canada
Mahmoud Mamou and Youssef Mébarki
National Research Council, Ottawa, Ontario K1a 0r6, Canada

Abstract: The objective of this research work is the development of an actuation control concept for a new morphing wing mechanism using smart materials made of Shape Memory Alloy (SMA) for the actuators and surface mounted pressure sensors. These actuators modify the upper wing surface, made of a flexible skin, so that the laminar-to-turbulent transition point moves close to the wing airfoil trailing edge. An intelligent controller was used for the open loop development step of the morphing wing project, and the closed loop of the morphing wing system included, as an internal loop, the controller actuation lines, based on the pressure information received from the sensors and on the transition point position estimation. The strong nonlinearities of the SMA actuators' characteristics and the system requirements led to the choice of a Fuzzy Logic Controller. The input-output mapping of the fuzzy model was designed taking account of the system's error and change in error. After preliminary numerical simulations using to tune the controller, an experimental validation was performed. The experimental validation consisted in experimental bench tests in laboratory conditions in the absence of aerodynamic forces, and in wind tunnel tests; the transition point real time position detection and visualization were realized.
2010 Mathematics Subject Classification: 12 Field theory and polynomials; 65 Numerical analysis; 93 Systems theory; control
Keywords: Laminar-to-turbulent transition; Morphing wing; Control System; Smart material actuators; Wind tunnel.

1. Introduction

The recent aims in aviation bring in front new horizons for researches related to the drag reduction through morphing an adaptive wing, which is motivated by rising fuel costs and environmental concerns. The concept relies on delaying the transition location toward the wing trailing edge by morphing the upper surface of the wing. The main objective of this concept is to promote large laminar regions on the wing surface, thus reducing drag over an operating range of flow conditions characterized by Mach numbers, airspeeds, and incidence angles [1]-[3]. The airborne modification of an aircraft wing airfoil shape can be realized continuously to maintain laminar flow over the wing surface as flight conditions change.

To achieve such a full operating concept, a closed loop control system has to be developed to link the flow fluctuations over the wing surface to the morphing mechanism (actuators).

The flow fluctuation signals can be detected by conventional pressure transducers or the new emerging pressure optical sensors.

Linked to a controller system, the collected data would be treated in real time aiming to identify the location of transition and then sending a signal to the actuator system to adjust the wind surface to delay the transition location.

The here presented work was performed under the 7.1 Consortium for Research and Innovation in Aerospace in Quebec (CRIAQ) collaborative project between academia and industries. In this project, the laminar flow behavior past aerodynamically morphing wing was improved to obtain significant drag reductions.

The project partners were the École de Technologie Supérieure (ETS), École Polytechnique of Montreal, the Institute for Aerospace Research at the National Research Council Canada (IAR-NRC), Bombardier Aerospace, and Thales Avionics.

This collaboration calls for both aerodynamic modeling as well as conceptual demonstration of the morphing principle on real models placed in the wind tunnel.

[1] Tel.: (514) 396-8560; Fax: (514) 396-8595
E-mail address: ruxandra.botez@etsmtl.ca; *URL:* http://www.etsmtl.ca

Preconditioned Iterative Solvers for Multi - Component Convection – Diffusion – Reaction Equations

Gheorghe Juncu[*], Aurelian Nicola[#], Constantin Popa[**], Elena Stroila[***]

[*]*Politehnica University of Bucharest, Romania;*

[#]*Ovidius University of Constanta, Romania*

[**]*Ovidius University of Constanta and "Gheorghe Mihoc-Caius Iacob" Institute of Statistical Mathematics and Applied Mathematics, Bucharest, Romania*

[***]*Research Center for Navy, Constanta, Romania*

Abstract: Computational fluid dynamics calculations of reacting flows or flows in which multiple species mix find widespread use in a large number of practical applications (combustors, catalytic converters, mixing devices in the chemical industry and in biological applications, etc). Usually, mass diffusion in multi-component mixtures is modeled by Fick's law for binary systems. The use of Fick's law for binary diffusion to model species diffusion in multi-component mixtures with more than two species is inappropriate because it does not ensure that the sum of the species diffusion fluxes equals zero. For these reasons, the need of efficient numerical schemes to solve diffusion – convection – reaction equations for multi-component systems is well recognized. The aim of the present work is to analyze the numerical performances of the preconditioned conjugate gradient algorithms (CGLS and GMRES) in solving multi-component, linear, convection - diffusion – reaction mass transfer equations, discretized with finite difference schemes. For the conjugate gradient algorithms, left and right preconditioning with LU (Cholesky) and incomplete LU (Cholesky) decomposition of the discrete Laplace operator was tested. The influence of the preconditioning on the mesh behaviour of the convergence rate was studied. The numerical results obtained show satisfactory performances.

2010 Mathematics Subject Classification: 80 Classical thermodynamics, heat transfer, 80A20, 80A32

Keywords: multi-component diffusion, convection-diffusion-reaction equation, sphere, Stokes flow, finite difference, conjugate gradient, preconditioning

1. Introduction

The mass transfer phenomenon in many real – life and engineering processes often govern the kinetics of the overall process. An accurate modeling of the mass transfer is therefore necessary, opportune and principally possible, particularly in multi-component systems. For these reasons, multi-component diffusion has gain a considerable increasing interest during the last decades.

For the description of mutual diffusion in multi-component systems, two approaches are prevalent in literature [1, 2]: generalized Fick's law and the Maxwell - Stefan theory.

The two formalisms are fully equivalent. A full presentation of these approaches is beyond the scope of this work. For more details we refer to [1 – 8].

Mathematical solutions for multi-component mass transfer in complex situations are not very plentiful. In many cases, idealized models, such as film model or the penetration model, were used.

From the computational point of view, the numerical solution of the multi-component diffusion equations is still a difficult, open problem (see Wangard et al. [9], Böttcher [10], Leonardi and Angeli [11] and the references cited herein). Additionally, even highly developed CFD software packages that are applied in engineering sciences do not contain the special features of the multi-component diffusion.

The aim of the present work is to analyze the numerical performances of the preconditioned conjugate gradient algorithms (CGLS and GMRES) in solving steady state multi-component mass transfer equations. The test problem is a linear convection - diffusion – reaction equation with four chemical species.

2. Basis Equations and Method of Solution

Consider the steady, axisymmetric, creeping flow of a Newtonian incompressible fluid past a rigid sphere. Oscillations and rotation of the sphere do not occur during the movement.

[*] Tel.: + 40 241 606 475, Fax: + 40 241 606 424
E-mail address: cpopa@univ-ovidius.ro *URl:* http://csam.univ-ovidius.ro/~cpopa

Criteria and applications of absolute stability in the automatic regulation of some aircraft course with autopilot

Mircea Lupu[1*], Olivia Florea[**], Ciprian Lupu[***]
*Transilvania University of Brasov, Faculty of Mathematics and Computer Science, m.c. A.O.S.R.
**Transilvania University of Brasov, Faculty of Mathematics and Computer Science
***Politehnica University of Bucharest, Faculty of Automatic and Computer Science

Abstract: In this paper are presented the methods of study of the automatic regulation of the absolute stability for some nonlinear dynamical systems. Two methods for the absolute stability are specified: a) the A.I. Lurie method with the effective determination of the Lyapunov function; b) the frequencies method of the Romanian researcher V.M. Popov that uses the transfer function in the critical cases. The authors develop a new sufficient criterion of absolute stability, with efficient technique of calculus.
Using this theoretical support are present two applications with analytical solutions:
1. the absolute stability of the rate of aircraft (airplane) equipped with autopilot
2000 Mathematics Subject Classification: 34D20, 34D23, 37N35
Keywords: absolute stability, Lurie methods, the automatic control, frequency method, automat pilot

1. Introduction

The automatic regulation for the stability of dynamical systems occupies a fundamental position in science and technique, following the optimization of the technological process of the cutting tools, of the robots, of the movement vehicles regime or of some machines components, of energetic radioactive regimes, chemical, electromagnetic, thermal, hydro-aerodynamic regimes, etc.

The studies and the technical achievements are complex by mathematical models for closed circuits with input - output, following for the automatic regulation the integration of some mechanisms and devices with inverse reaction of response for the control and the fast and efficient elimination of the perturbations which can appears along these processes or dynamical regimes. Generally these dynamical regimes are nonlinear and it was necessarily some contributions and special achievements for automatic regulation, generating the automatic regulation of absolute stability (a.r.a.s.) for these classes of nonlinearities.

We highlight two special methods (a.r.a.s.):

- Liapunov's function method discovered by A.I. Lurie [13,15,20] and developed into a series of studies by M.A Aizerman, V.A. Iakubovici, F.R. Gantmaher, R.E. Kalman, D.R. Merkin [14] and others [1,17].
- Frequency method developed by researcher VM Popov [18] generalizing the criterion of Nyquist, then developed in many studies [1,2,15].

We note the contributions of Romanian researchers recognized by the works and monographs on the stability and optimal control theory: C. Corduneanu, A. Halanay, V. Barbu, Th. Morozan, G. Dinca, M. Megan, Vl. Rasvan, V. Ionescu, M.E. Popescu, S. Chiriacescu, A. Georgescu and also who studied directly on (a.r.a.s.): I. Dumitrache [4] D. Popescu [16], C. Belea [2], V. Rasvan [19], S. Chiriacescu [3] and other recent works [6,…,12].

The research has shown that both methods are equivalent, and studies can be qualitatively or numerically. In this paper we presented the actual making methods in cases of singularity studies across applications.

2. (A.R.A.S.) Using the Liapunov's function method

In this part we'll present the Lurie's ideas and the effective method for found the Liapunov's function [13,14,2,19]. Generally, the systems of automatic regulation are composed from the controlled processor system, and sensory elements of measurement, acquisition board, and the mechanism feedback controller. The regulator will mean all the sensors and the acquisition board, but the controller is included feedback mechanism. Parameters characterizing the object control system to control work mode are measured by sensors, and their

[1] *E-mail-address*: m.lupu@unitbv.ro
[**] *E-mail-address*: olivia.florea@unitbv.ro
[***] *E-mail-address*: cip@indinf.pub.ro

Analysis of external and internal unsteady flows with oscillating boundaries at low Reynolds numbers

Dan Mateescu*
Doctor Honoris Causa, FCASI, AFAIAA, Erskine Fellow
Professor, Aerospace Program, Mechanical Engineering Department McGill University, Montreal, QC, Canada

Abstract: This paper presents solutions for external flows past airfoils executing oscillatory motions at low Reynolds numbers and for unsteady internal flows in confined configurations with oscillating boundaries and variable inflow velocity. These unsteady flow solutions have been obtained for the first time by this author and his graduate students. Both the internal and external unsteady flow problems are solved in fixed computational domains, obtained from the physical domain using time-dependent coordinate transformations, in which the moving boundary conditions are efficiently and rigorously implemented. The unsteady flow solutions are obtained in the fixed computational domain with efficient numerical methods developed by the author, based on the time-accurate integration of the Navier-Stokes equations. This method, which does not require a lengthy grid generation procedure, is second-order-accurate in both time and space; it uses a second-order three-point-backward implicit scheme for the real time discretization, followed by a pseudo-time relaxation procedure using artificial compressibility and a factored ADI scheme for the pseudo-time integration. A second-order central finite differencing formulation is used on a stretched staggered grid. A special decoupling procedure using the continuity equation reduces the problem to the solution of scalar tridiagonal systems of equations, which enhances substantially the computational efficiency of the method. The paper presents also a thorough analysis of the unsteady flow separations at low Reynolds numbers and studies the influence of various flow parameters (such as the Reynolds number and the amplitude and frequency of oscillations) on the aerodynamic characteristics and on the flow separations.

2000 Mathematics Subject Classification: 76, 76G25, 76D05, 76M20, 65M06, 35Q30
Keywords: Unsteady Flows, Low Reynolds number, Subsonic Aerodynamics, Computational Aerodynamics, Viscous Flows

1. Introduction

The study of external and internal flows at low Reynolds numbers recently received a special interest.

For external aerodynamics of airfoils and wings at low Reynolds numbers, the interest is driven by a variety of applications ranging from domestic windmills to special military aircraft and unmanned aerial vehicles (UAV), which were made possible by the recent advances in the micro-electro-mechanical systems (MEMS). Very small aircrafts called micro-air-vehicles (MAV) can operate in various indoors or outdoors environments including tunnels, desert and jungle (see Davis *et al*. 1996 and Mueller 1999, who presented a detailed review of the MAV concept). For a small size micro-air-vehicle, with a mass between 10 and 20 grams and flying at very low speeds, the Reynolds number could be about 1000 or even lower. For internal flows in confined configurations, the interest is driven by a variety of industrial applications ranging from the cooling flows in electronic devices [Velasquez *et al*. 2008] to the flow modeling the continuous casting operation to near-net shape [Gerber *et al*. 2007]. The airfoil aerodynamics at very low Reynolds numbers, between 400 and 6000 is dominated by viscous effects and flow separation phenomena, and is very challenging and different from those of conventional aircraft. Several authors [Kunz & Kroo 2000] found that many successful aerodynamic codes developed for the normal range of the Reynolds number are not well suited for very low Reynolds numbers. Several studies have been published on the airfoil analysis in steady flow at very low Reynolds numbers. Thus, [Kunz & Kroo 2000] used in their computational study of low Reynolds number flows (between 1000 and 6000) the INS2D code based on an upwind finite differencing scheme developed at NASA Ames by Rogers & Kwak [1990]. Also, Mateescu and Abdo [2010, 2004] studied the steady flows past airfoils at very low Reynolds numbers between 400 and 6000 with a novel method using artificial compressibility and a central finite difference formulation on a stretched staggered grid. Recently, Mateescu *et al*. [2011] presented new solutions for unsteady flows past oscillating airfoils at low Reynolds numbers, obtained with a time-accurate method to solve the Navier-Stokes equations developed by the authors.

* Tel: 514 398-6284 (Canada)
E-mail address: dan.mateescu@mcgill.ca

New Hybrid Solutions for Compressible Navier – Stokes Partial – Differential Equations

Adriana Nastase*

Aerodynamics of Flight, RWTH, Aachen University, 52062 Aachen, Germany

Abstract: The new developed, hybrid, meshless solutions for the three-dimensional compressible Navier-Stokes layer (NSL) are presented here in improved form. These NSL solutions use analytical potential solutions of the flow on the same flying configurations (FCs) twice, namely: at the NSL's edge (instead of parallel flow used by Prandtl in his boundary layer theory) and in the structure of the velocity's components, which are expressed inside the NSL, as products between the corresponding potential velocity's components with polynomes with arbitrary coefficients, versus a spectral variable. These coefficients are used to satisfy the NSL's partial differential equations (PDEs), in an arbitrary chosen number of points. The proposed reinforced numerical NSL's solutions are split, have important analytical properties, like correct last behaviors, correct jumps along the singular lines like subsonic leading edges, junction lines wing/fuselage and wing/leading edge flaps, are accurate and rapid convergent. These hybrid NSL's solutions are useful for the computation of the friction drag coefficients of FCs, for the qualitative analysis of NSL's behaviors in vicinity of its critical lines, for the global optimal design of the FC's shape, etc.

2010 Mathematics Subject Classification: 35L70 Partial differential equations, non-linear, second order PDEs of hyperbolical type; 76N15 Compressible fluids and gasdynamics; 76D05 Fluid mechanics, Navier-Stokes PDEs; 76N20 Boundary layer theory; 76N17 Viscous-inviscid interaction

Keywords: Three-dimensional compressible Full Navier-Stokes PDEs; Meshless, hybrid analytical-numerical solutions; Three-dimensional compressible and viscous flow; Qualitative analysis of the behaviors of the elliptic and hyperbolic PDEs

1. Introduction

The study starts from the PDEs of the three-dimensional compressible stationary full NSL, as given in [1], [2], namely,

- the PDE of continuity

$$\frac{\partial(\rho\ u_\delta)}{\partial x_1}+\frac{\partial(\rho\ v_\delta)}{\partial x_2}+\frac{\partial(\rho\ w_\delta)}{\partial x_3}=0 \quad ; \tag{1}$$

- the PDEs of impulse

$$u_\delta\frac{\partial u_\delta}{\partial x_1}+v_\delta\frac{\partial u_\delta}{\partial x_2}+w_\delta\frac{\partial u_\delta}{\partial x_3}=\frac{1}{\rho}\left\{-\frac{\partial p}{\partial x_1}+\frac{\partial}{\partial x_1}\left[\mu\left(2\frac{\partial u_\delta}{\partial x_1}-\frac{2}{3}\ div\,\vec{V}\right)\right]+\frac{\partial}{\partial x_2}\left[\mu\left(\frac{\partial u_\delta}{\partial x_2}+\frac{\partial v_\delta}{\partial x_1}\right)\right]+\frac{\partial}{\partial x_3}\left[\mu\left(\frac{\partial w_\delta}{\partial x_1}+\frac{\partial u_\delta}{\partial x_3}\right)\right]\right\}, \tag{2a}$$

$$u_\delta\frac{\partial v_\delta}{\partial x_1}+v_\delta\frac{\partial v_\delta}{\partial x_2}+w_\delta\frac{\partial v_\delta}{\partial x_3}=\frac{1}{\rho}\left\{-\frac{\partial p}{\partial x_2}+\frac{\partial}{\partial x_2}\left[\mu\left(2\frac{\partial v_\delta}{\partial x_2}-\frac{2}{3}\ div\,\vec{V}\right)\right]+\frac{\partial}{\partial x_3}\left[\mu\left(\frac{\partial v_\delta}{\partial x_3}+\frac{\partial w_\delta}{\partial x_2}\right)\right]+\frac{\partial}{\partial x_1}\left[\mu\left(\frac{\partial u_\delta}{\partial x_2}+\frac{\partial v_\delta}{\partial x_1}\right)\right]\right\} \tag{2b}$$

* Tel.: 0049 241 809 5438; fax: 0049 241 8092 173
E-mail address: nastase@lafaero.rwth-aachen.de

Calculation of transonic and supersonic internal flows using upwind schemes

Mihai Leonida Niculescu*, Sterian Dănăilă**
** Ph. D., National Research Institute for Aerospace Research "Elie Carafoli" INCAS, 220 Iuliu Maniu Avenue, 061126 Bucharest 6, Romania, P.O. 76*
*** Professor, Faculty of Aerospace Engineering, Politehnica University of Bucharest, 1 Polizu, sect. 1, 011061, Bucharest, Romania*

Abstract: A study of some popular upwind schemes applied to transonic internal flows using some well-known test cases is done in this paper. We focused on upwind schemes because the central space discretizations have symmetry with respect to a change in sign; therefore, the physical propagation of perturbations along characteristics, typical of hyperbolic equations is not considered in the definition of numerical model. In contrast to the central space discretizations, the upwind schemes whose origin may be due to Courant et al. [2] are directed towards the introduction of the physical properties of the flow equations into the discretized formulation that has leads to upwinding techniques such as flux vector splitting and flux difference splitting. In order to test the accuracy, robustness and efficiency of some popular upwind methods (van Leer scheme, Roe scheme and Liou's AUSM$^+$ scheme); we used some well-known test cases.

2010 Mathematics Subject Classification: 65 Numerical Analysis; 76 Fluid mechanics
Keywords: upwind schemes; transonic and supersonic flows; bump; CFD

1. Introduction

Development of accurate and efficient numerical schemes for compressible flow governing equations is essential due to increasing engineering demand for aircraft and gas turbines design. Such schemes are particularly important when aircraft or gas turbine aeroelasticity is simulated using a fully coupled fluid/structural model that needs huge computational resources. For this reason, an efficient and robust upwind scheme capturing properly shock waves, contact surface discontinuities, and near-wall region is very desirable.

It is well known that the introduction of physical properties in the upwind schemes for the Euler equations can be done at different levels.

The first level introduces only information on the sign of the eigenvalues; therefore, the flux terms are split and discretized directionally according to the sign of the associated propagation speeds. This leads to the flux vector splitting (FVS) methods such as Steger-Warming scheme [20, 10, 3], van Leer scheme [11, 10, 3], Liou's AUSM$^+$ scheme [14], schemes developed by Zha et al. [26, 27, 28] etc.

A higher level of introduction of physical properties into the construction of the scheme can be defined following the very remarkable scheme of Godunov [8]. In this method, the conservative variables are considered as piecewise constant over the cells at each time step and the time evolution is given by the exact solution of the Riemann (shock tube) problem at the inter-cell boundaries; therefore, the properties derived from the exact local solution of the Euler equations are introduced in the discretization.

Unfortunately, for the multidimensional cases, solving the local Riemann problem is very expensive from the computational point of view. Hence, Roe [18] and Osher et al. [5, 6, 16] developed schemes that solves approximately the Riemann problem and their methods are called sometimes as flux difference splittings (FDS). In general, the FDS methods in comparison with the FVS methods are more accurate and more expensive from the computational point of view.

2. Riemann problem

For shock tube problems, the interests are focused on: 1) the quality (monotonicity and sharpness) of the shock wave and contact discontinuity 2) entropy condition in order to avoid the expansion shock 3) the maximum allowable CFL to be used for explicit Euler time marching method.

The Sod shock tube problem [19] modified by Toro [22] is the first test case, which consists of a one-dimensional Riemann problem with the following initial parameters, for left and right states of an ideal gas:

* *Tel: +4021 434 00 78; fax: +4021 434 00 82
E-mail address: mniculescu@incas.ro; *URL:* http://www.incas.ro
** *E-mail address:* sterian.danaila@upb.ro

Mathematical and numerical modeling of a vortex ring

Cornelia Niță*, Sterian Dănăilă**
Department of Aerodynamics, "Elie Carafoli" National Institute for Aerospace Research – INCAS, 061126, Bucharest, Romania
Department of Aerospace Sciences, University „Politehnica" of Bucharest – 011061, Bucharest, Romania

Abstract: This paper deals with mathematical and numerical modeling of a vortex ring motion in complex geometries. The vortex ring is injected in the domain where the fluid is in repose. The immersed boundary method (IBM) with "ghost cell" and the direct forcing method are used for a geometry consisting of a square body in a flow. The direct forcing method has already been successfully applied by P. Orlandi, R. Verzicco *et al.* [13, 8], where the interior of the obstacle is calculated in the same way as the remaining of the computation region because the solution of equation for pressure correction is obtained with the efficient fast Poisson solver. In this paper, the "ghost cell" method was implemented and in order to solve the pressure correction equation and for implicit formulation of viscous dissipation in momentum equations, the Stone's method (SIP) was used. The SIP method, "strongly implicit procedure", eliminates the restrictions that appear in the fast Fourier solvers (e.g. the subroutine BLKTRI from FISHPACK library).The Direct Numerical Simulation (DNS) are used for a detailed analysis of vortex ring interaction with the wall and with the obstacle and to compare numerical results with experimental ones.
2010 Mathematics Subject Classification: 65 Numerical Analysis, 76 Fluid mechanics
Keywords: vortex ring, immersed boundary methods, incompressible fluid, strongly implicit method

1. Introduction

The fundamental and practical interest of scientific community for vortex rings simulations has generated a large volume of literature in the last decade.

The vortex rings have long fascinated scientists by their behavior and especially by their compact nature, in this manner Saffman [11] shows that: "the formation (of vortex ring) is a problem of vortex sheet dynamics, the steady state is a problem of existence, their duration is a problem of stability, and if there are several we have a problem of vortex interactions".

The penetration of the jet in the domain, in which fluid is in repose, will engage vorticity in the form of a vortex. This is the phase of formation of vortex ring. After the injection is condemnatory the vortex ring is detached from the generator and moves downstream by its own speed. It grows by viscous diffusion and entrainments the ambient fluid. This is the phase of post-formation. Kelvin-Helmoltz instabilities will be produced if the injection is long term. This type of instabilities is more complex phenomena of interaction between vortices.

The study of interactions between vortex structures and solid walls allows understanding the generation of vorticity at wall. This is important in several practical applications when the new vorticity plays a decisive role in vortex dynamics near the surfaces which determining the forces on the body. By smoke visualization method, for example, was difficult to capture the small vortices produced on the cylinder's walls. On the other hand, the numerical simulations allow not only the distinction between primary, secondary and tertiary vortices and also their interactions.

The vortex ring appears in different forms in practical applications and in nature, for example at injection in internal combustion, at propulsion and locomotion of some animals (jellyfish), at volcanic eruptions or when the blood is pumped from left atrium to left ventricle across open mitral valve in human heart.

The instability of vortex ring was studied by Krutzsch [1], who was the first that offered pictures and data which describes the phenomena, but the full process of instability, growth and transition was described by the investigation of Crow [2] into aircraft wakes.

Maxworthy [5] and Widnall & Sullivan [16] confirmed experimentally some of Krutzsch's observations and proved that the instability consists of the growth of azimuthal waves and their wavenumber depends of the

---

* Tel.: +40 21 434 0083; fax: +40 21 434 0082
E-mail address: nitac@incas.ro ; *URL:* http://www.incas.ro
** Tel.: +40 21 402 3967; fax: +40 21 318 1007
E-mail address: sterian.danaila@upb.ro ; *URL:* http://www.upb.ro

Flow control through an airplane's intake

Andreea Cristina Petcu*

INCD Turbomotoare COMOTI, B-dul Iuliu Maniu nr. 220D, sector 6, cod 061126, OP 76, CP174, Bucuresti, Romania

Abstract: In this paper is presented a study regarding the control of air flow. For this purpose it was considered the study of the air flow through a supersonic airplane’s intake. Aircraft engine intakes for upersonic flight speeds have sharp edged inlet cowls, and in consequence considerable losses in total pressure of the entering air occur at take-off conditions. To reduce this loss we opened an auxiliary air inlet in the side of the intake. Our purpose was to find a relationship between the medium total pressure at the entrance in the compressor and the dimension of the auxiliary inlet knowing that at the compressor’s entrance the Mach number of the air flow must be 0.5. The air flow numerical simulations are realized using FLUENT.
2010 Mathematics Subject Classification: 76 Fluid mechanics
Keywords: flow control, intake, take-off conditions, inviscid fluid, compressible fluid, total pressure

1. The intake device

The air intake is that part of an aircraft structure by means of which the aircraft engine is supplied with air taken from the outside atmosphere. The air flow enters the intake and is required to reach the engine face with optimum levels of total pressure and flow uniformity. These properties are vital to the performance and stability of engine operation [1].

The main conditions to be met by the intake device are [2]:

- To ensure at the entry into the compressor a uniform distribution of speeds in time and space; thus there shouldn’t be any pulse, local turbulence or variations of speed in radial and angular directions. A uniform distribution of speeds will generate a uniform distribution of static and total pressures, condition for stable operation of the compressor;
- To partly transform the kinetic energy of the air flow into mechanical work of compression;
- The evolution of the air into the intake device to generate a minimum possible total pressure drop, for a 1% decrease in pressure results in a decrease in traction force of 0.5-1.2% depending on the engine design solution and flight regime;
- The plane’s drag induced by the intake device to be as little as possible.

Therefore an optimal intake device cannot be achieved for all flight regimes, but a compromise solution can be developed corresponding to the main flight regimes depending on the type of aircraft used.

The intake device constructive solution depends mainly on the flight speed of the aircraft (subsonic or supersonic regime), as well as the location of the engine (fuselage or external gondola). If the engine is embedded in the fuselage the air intake stands outside, while the intake’s sewerage is a constructive element of the plane’s fuselage. In this case the total pressure loss may depend on the constructive solution of the intake’s sewerage, which varies from plane to plane, and the drag is determined only by the air intake. If the engine is located on an external gondola the admission device is part of the engine, whose constructive solution must be analyzed in terms of maximum performance of the entire propulsion system.

The air’s velocity leaving the intake device is subsonic. For example at turbo-reactors the axial component of the velocity at the compressor’s entry is between 120-160 m/s in the case of centrifugal compressors and 150-220 m/s in the case of axial compressors.

After the flight speeds of the plane for which the air intake was designed we distinguish [3]:

1. $M < 0.7$ – subsonic admission device, in any point of the flow the critical regime is not reached.
2. $M \in (0.7-1.2)$ - admission device with one normal shock wave (Pitot type).
3. $M \in (1.2-1.5)$ - supersonic admission device with one oblique shock wave and one normal shock wave.
4. $M \in (1.5-2.5)$ - supersonic admission device with two oblique shock waves and one normal shock wave.

* Tel: 0214340240, fax: 0214340241
E-mail adress: andreea.petcu@comoti.ro; *URL*: http://www.comoti.ro

The intelligent pulsatory liposome

Dumitru Popescu*, Alin Gabriel Popescu**

*Department of Mathematical Modeling 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 Anatomy, Animal Physiology and Biophysics, Center of Neurobiology and Molecular Physiology, Faculty of Biology, University of Bucharest, Splaiul Independenței 91–95, Bucharest 050095, Romania
**Department of Computer Sciences, IT CORE SRL, Str. Garoafei, nr.10, sector 5, Bucharest-051235, Romania

Abstract: In this paper, we have considered the problem of the pulsatory lipid vesicle. Under positive osmotic stress a lipid vesicle swells up to a critical size, when suddenly a transbilayer pore appears. A part of the intracellular material leaks out through this transmembrane pore and the liposome membrane relaxes and finally, it recovers. In certain conditions, the vesicle swelling start again and its evolution will be a cyclic process. We have obtained the differential equations of both the vesicle and the pore dynamics. Also, we have analyzed characteristic parameters of the first cycle of a pulsatory lipid vesicle (swelling time, pore lifetime, material quantity leaked out during a cycle). Finally, we present the condition to programme an n-cycles working vesicle, so that to be an intelligent liposome. Such a liposome may be used for special substances dosage at microsites hardly to reach.

2010 Mathematics Subject Classification: 92 Biology and other natural sciences
Keywords: Pulsatory liposome; Dynamical parameters; Usefull parameters

1. Introduction

The pore appearance in lipid bilayers following some controlled processes may be an usefull and interesting way for transmembrane transport [3, 5].

The pore appearance in plane lipid bilayer may be influenced by thickness fluctuations [4, 5, 8] or by structural defects [4, 6, 7].

In the lipid vesicle the pore appearance, may be favored by mechanical tension induced by different ways [10]. 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]. There are two very interesting biotechnological applications which request the increase of membrane permeability: gene therapy and targeted special substances delivery [12].

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

In this paper, we will make an analysis of the two stages of a cycle from working life of a pulsatory liposome: swelling and relaxation. Then, we will the dynamic parameters characterizing a cycle of the periodical activity of a pulsatory vesicle. Before the two part highlighted before, we placed a subchapter which contains a description of the phenomenological base of the running of a pulsatory liposome.

2. Phenomenological base of a pulsatory liposome

Let us consider a liposome filled with aqueous solution containing an osmotic solute. The initial state of the liposome is characterized by smooth and unstretched lipid membrane and by the internal solute concentration. It is considered as equilibrium reference state. This liposome is inserted into a bath with a hypotonic aqueous medium. Such, the reference state become the initial state of the liposome dynamics. Due to osmotic pressure, created by the transmembrane gradient of solute concentration, water molecules inflow inside to liposome, across its membrane.

The osmotic flow of solvent determines: 1) the swelling of the liposome; 2) the stretching of liposome membrane; 3) the dilution of the internal solution. Also, the surface tension increases in the same time with the liposome expansion. The surface tension increases the pressure inside the vesicle, while the osmotic pressure decreases.

* Tel.: +40 21 318 2433; fax: +40 21 318 2439
E-mail address: popescu1947@yahoo.com

Regarding the perturbed operating process of DB propellant rocket motor at extreme initial grain temperatures

Doru Safta*, Titică Vasile*, Ioan Ion**
*Military Technical Academy – MTA, 050141, Bucharest, Romania
**University "Eftimie Murgu", 320085, Resita, Romania

Abstract: Despite many decades of study, the combustion instability of several DB propellants is still of particular concern, especially at extreme grain temperature conditions of rocket motor operating. The purpose of the first part of the paper is to give an overview of our main experimental results on combustion instabilities and pressure oscillations in DB propellant segmented grain rocket motors (SPRM-01, large L/D ratio), working at extreme initial grain temperatures. Thus, we recorded some particular pressure-time traces with significant perturbed pressure signal that was FFT analysed. An updated numerical model incorporating transient frequency-dependent combustion response, in conjunction with pressure-dependent burning, is applied to investigate and predict the DB propellant combustion instability phenomenon. The susceptibility of the tested motor SPRM-01 with DB propellant to get a perturbed working and to go unstable with pressure was evidenced and this risk has to be evaluated.
In the last part of our paper we evaluated the influence of recorded perturbed thrust on the rocket behaviour on the trajectory. The study revealed that at firing-table initial conditions, this kind of perturbed motor operating may not lead to an unstable rocket flight, but the ballistic parameters would be influenced in an unacceptable manner.
2010 Mathematics Subject Classification: 70 Mechanics of particles and systems; 76 Fluid mechanics; 80 Classical thermodynamics, heat transfer
Keywords: Thermodynamics of rocket propulsion; Steady and unsteady flows; Unsteady combustion; Combustion instability; Pressure-coupled response function; Flight Dynamics

1.1. Introduction

As a rocket in-flight is exposed to extreme temperatures, from 220K up-to 340K, it is very important to see how the solid propellant performs in these extreme temperature variations.

The first part of the paper is focused on the analyse of our main experimental results on combustion instabilities and pressure oscillations in the solid rocket motors (SPRM-01, large L/D ratio) with Double Base (DB) propellants. We took into account the main differences between DB propellants and their modern successor such as their homogeneous nature, very predictable and constant burn rate and smokeless exhaust products, which has made this type of propellants very tactically attractive for military applications. The DB solid propellant motors usually used for small and medium tactical missiles are characterized by high pressure levels, short burning times and rather complex grain shapes.

Combustion instability is a major concern in all the propulsion systems and it is characterized by vibrations within the combustion chamber, generally measured as an oscillating pressure. Thus, pressure oscillations internal to solid rocket motors have received much attention in the propulsion community.

The perturbed operating process especially at extreme initial grain temperatures is a potential problem in the development of any new solid rocket motor and it can appear even after the start of production.

In our researches we recorded some particular pressure-time profiles at extreme ambient temperatures (233; 313) K with significant perturbed pressure signal that was FFT analysed.

The classical linearized theory of unsteady combustion of homogeneous propellants and energetic materials is based on the assumption of quasi-steady (QS) reaction zones in both the gas and condensed phases. This theory was developed on the basis of the flame modelling (FM) approach or of the phenomenological Zeldovich-Novozhilov (ZN) approach.

* Tel.: +40 72 222 5866; fax: +40 21 410 2507
E-mail address: doru.safta@yahoo.com; *URL:* http://www.mta.ro

New solution for sonic boom mitigation. Concept and testing methodology. Application at European supersonic business jet

Constantin Sandu*, Dan Braşoveanu**
* Project Head and Manager–SC Turbomecanica SA/ SMCPFA, Bucharest, Romania
** Senior scientist-CSC, Washington, USA

Abstract: At present, two research directions are practically in attention of researchers: new supersonic aircraft design using classic technology and shaping technology.
The present paper presents a new solution (method) that could be applied for sonic boom mitigation.
According to this method, shock wave is scattered (dispersed) at source by means of surface vibration. Special membranes made of Kevlar cover the aircraft nose, wing and tail leading edges. A variable pressure applied in a liquid film induces membrane vibration.
When membrane vibrates, angle α between air stream and aircraft local surface is periodically changed with a certain frequency, ν .
It can be demonstrated that if Mach number is in the range 1…2, a small variation $\Delta\alpha$ of cone/wedge angle produces large variation of shock wave angle, $\Delta\beta$. Thus, at the ground level, the N wave should be spread on a much larger area than in the natural case.
2010 Mathematics Subject Classification: 76 Fluid mechanics;
Keywords: shock wave mitigation; supersonic business jet

1. The present state of the art

The Defense Advanced Research Projects Agency (DARPA) selected in 2002 the performers for a Phase II of the Quiet Supersonic Platform (QSP) program [1].

The selected companies were:

- Lockheed Martin, Advanced Development Company, Palmdale, Calif.;
- Northrop-Grumman Corporation, El Segundo, Calif.;
- Arizona State University, Tempe, Ariz.;
- General Electric, Cincinnati, Ohio.

For about 8 900 000$ these system integrators updated their aircraft/engine designs and technology assessments based on revised program goals, perform validation of their designs, perform utility and cost analysis, and develop technology maturation roadmaps.

Northrop-Grumman Corporation received additional funds to conduct flight demonstration of direct sonic boom mitigation using a modified F-5E aircraft.

The aircraft was modified using a special designed nose glove to produce a shaped sonic boom profile at the ground.

Before flight demonstration, wind tunnel test validated the computed sonic boom signature predictions for the modified F-5E, and finally a series of flight tests validated the predicted persistence of shaped sonic booms.

The program demonstrated for the first time that an appropriately shaped aircraft can produce a mitigated sonic boom.

Some design solutions are presented in figs 1, 2 and 3.

* Tel.: +04 0723147322; fax: -
E-mail address: constantin.sandu@turbomecanica.ro; URL: http://www.turbomecanica..ro

A mathematical model to study the horizontal oscillations of the railway vehicles

Ioan Sebeşan*, Dan Băiaşu**, Gheorghe Ghiţă***
**Faculty of Transports, Politehnica University Bucharest,Bucharest, Romania*
***Atelierele CFR Grivita S.A., Calea Grivitei 359, 010178, Bucharest, Romania*
****Romanian Academy Institute of Solids Mechanics – IMSAR, Bucharest, Romania*

Abstract: The article presents a mathematical model for the study of a passenger coach hunting motion using the multibody approach. The model comprises the lateral displacement, rolling and yawing motions for the main constitutive elements: axles, bogies and case. The equation system is written applying energetic methods. The forced vibrations determined by the irregular profile of the tracks are considered. The wheel – rail contact forces are expressed using the creepage coefficients established according to Kalker's linear theory. The equations system is solved through numeric methods using specialized calculus programs. The response of the system – passenger coach on a tangent track and the critical speed are determined.
2010 Mathematics Subject Classification: 34 Ordinary differential equations, 65 Numerical analysis, 70 mechanics of particles and systems
Keywords: multibody system model; mathematical model; mechanical system response; system lateral stability

1. Introduction

The lateral railway vehicle dynamics represent a study area of great interest in the actual context where more and more railway administrations implement the high speed trains, which prove to be efficient, economic and ecological transportation means. Trains circulating with speeds higher than 160 km/h generate vibrations in the vehicle body that induce significant operation problems: running instability, poor ride quality and track wear. From this point of view, an adequate design of the railway vehicles' suspensions holds an important role in maintaining the comfort and safety parameters of trains circulation.

Kinematic theoretical studies of the rolling apparatus' elements lateral and yawing motions [1], [2] have highlighted that the oscillation frequency grows proportionally to the circulation speed. The speed value where the amplitude of the oscillations grows and the vehicle movement becomes unstable is called critical speed. Starting off with this approach, various studies on the railway vehicle's lateral stability have showed the existence of two sources of instability for the railway vehicle:

- the bogie instability, induced by the axles' movement instability;
- the instability of the case, which appears when, in the low frequency domain, the vehicle body has the tendency of moving along with the bogie.

The dynamic behavioral study of the railway vehicle has two directions:

- the dynamic response of the system: simulating dynamic behavior due to external stimuli, determining the concentrated mass accelerations and speeds, and implicitly the forces that act upon the vehicle;
- the dynamic stability: the study over the system's stability in various operation conditions.

The mathematical modeling of the rail or of the railway vehicle is frequently used for study or in order to observe the rail's and railway vehicle's interaction with the tracks. The dynamic interaction between the vehicle and the tracks varies depending on the operation conditions, geography, the wheel rolling tread and rail and the weather conditions. Obtaining a mathematical model for the study of the dynamic behavior in the case of the railway vehicle implies to consider the latter being formed out of rigid bodies inter-connected through weightless suspension elements. Usually, for railway vehicles, the mathematical models consider the body case, the bogies and the wheelsets as concentrated mass. The equations system describing the movement of the mathematical model could thus have 42 quadratic coupled equations. Solving such a system represents a sometimes inconclusive undertaking regarding the vehicle's behavior. According to [1], [3], [4] for small amplitude movements, there is a relatively small connection between the vehicle's oscillations on a vertical and transversal directions, this is why some of the models presented in literature don't take into account the vertical oscillations in the study of movement on lateral direction or the horizontal oscillations for the study of vehicle vertical displacement. Starting with the 60's, numerous authors have dedicated studies to the lateral oscillations

** Tel.: +40 21 2240926; fax: +40 21 2241736
E-mail address: dan.baiasu@grivita.ro; *URL:*http://www.grivita.ro

Methods in dynamics of compass robotic arm

Ion Stroe*, Ştefan Staicu*, Andrei Crăifăleanu*
*"Politehnica" University of Bucharest, Department of Mechanics, Splaiul Independentei nr. 313, sala BN-01, cod 060042, Bucuresti, ROMANIA

Abstract: The principle of virtual work or Lagrange equations can be used in order to study the motion of a system under the action of external forces, or to find reactions, as well as various other internal forces. The principle of virtual work is used to study the motion of a system under the action of external and internal forces. Based on the principle of virtual work, the elaborated method solves the problem of calculus of input forces and joint forces of a compass robotic arm, sketched as a serial chain.
For the calculus of the internal forces, a new method based on Lagrange equations is presented. If an internal force has to be determined, a supplementary mobility is considered in the system. The internal force corresponding to the new mobility is found if zero mobility is imposed.
The general problems of the kinematics of systems and of the motion of rigid body systems with constraints are presented. Problems of kinematics are solved for constraints expressed by coordinates. Translation conditions and rotation conditions are analyzed. The motion of the system is considered in movable, non-inertial reference frame.
The application of the two methods, based on the principle of virtual work and on Lagrange equations, respectively, are illustrated by determining the bending moment in a Compass Robotic Arm. The geometry of this system is inspired from the European Robotic Arm (ERA).
The goal of the paper is not to present the European Robotic Arm, but to present methods of calculus. The geometry of the open loop mechanism is studied, but the methods can be also used for closed loops. Results regarding internal forces for the known motion of the robotic arm are obtained by numerical simulations.
The elaborated methods allow the solving of a large number of problems concerning the dynamics of systems of bodies.
2010 Mathematics Subject Classification: 70 Mechanics of particles and systems
Keywords: internal forces; internal torques; Lagrange formalism; virtual work

1. *Introduction*

Considering the gravitational effects, the relevant objective of the multi-bodies dynamics is to determine the input torques or forces and the external and internal joint forces. Several methods have been applied to formulate the dynamics, which could provide the same results concerning these torques or forces. The first one is using the Newton-Euler procedure, the second one applies the Lagrange formalism with its multipliers, while the third is based on the principle of virtual work [1], [4], [14].

Difficulties commonly encountered in dynamics of multi-bodies systems include problematic issues such as: complex spatial kinematical structures which possess a large number of passive degrees of freedom, dominance of inertial forces over the frictional and gravitational components and the problem linked to the real-time control and the solution of inverse dynamics.

In the first part of the paper, the application of the principle of virtual work is illustrated by determining several internal forces or torques in the joints of a Compass Robotic Arm.

In the second part, the method based on Lagrange equations is used to determine the bending moment in the same Compass Robotic Arm. In order to apply the method, first the general problem of the kinematics of rigid body systems with constraints is presented.

2. *Method based on the principle of virtual work*

2.1. *Kinematics analysis*

Let two bodies (T_i), (T_j) be with constrained motions by a coupling mechanism (Fig. 1). The motion of (T_i) with respect the inertial reference frame $O_0x_0y_0z_0(T_0)$ is determined by the position vector $\vec{r}_0^{\,C_i} = \overrightarrow{O_0C_i}$ of

*Tel.: +40 21 402 9250; fax: +40 21 411 53 65
E-mail address: craifaleanu@cat.mec.pub.ro; ycraif@yahoo.com; *URL:* http://cat.mec.pub.ro

New first integrals for the motion equation; The vortex equation; The continuity equation; Its first integral: the flow rate equation

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

Abstract: This work studies and clarifies some local phenomena in fluid mechanics, in the form of an intrinsic analytic study, regarding the motion, continuity and flow rate equations (for inviscid compressible fluids), and the vortex equation (for viscous incompressible fluids), and finding new first integrals. It continues a series of works presented at some conferences and at a congress in '08 – '10, representing a real deep insight into the still hidden theory of isoenergetic flow. Several new functions, surfaces and vectors were introduced: the polytropic integral surfaces, for the motion equation; Şelescu's incompressible roto-viscous vector, for the vortex equation; the 2-D "quasi-stream" function on the 3-D (**V**, **Ω**) surfaces, for the continuity equation; the surfaces of iso-normal mass flux density (over which the continuity equation of the steady flow of a compressible fluid in a thick stream tube admits a first integral, and whose envelope sheets are just the sections of uniform flow, if they exist), for the flow rate equation; the 3-D stream function vector, allowing new local and global forms for the continuity equation. A case of first integrability for the system of equations (motion and continuity) for the steady flow of an inviscid compressible fluid was also considered.
2010 Mathematics Subject Classification: 12 Field theory and polynomials; 31 Potential theory; 34 ODEs; 35 PDEs;70 Mechanics of particles and systems; 76 Fluid mechanics; 80 Classical thermodynamics, heat transfer
Keywords: conservative and biscalar vectors; rotational flows; steady and unsteady flows; inviscid and viscous fluids; compressible fluids; isentropic surfaces; polytropic surfaces; Şelescu's vector, space curves, zero-work surfaces (for some non-conservative vectors) and quasi-incompressible quasi-potential (Laplace's) lines (for a quasi-uniform rotational flow of an inviscid compressible fluid)

Presentation of problem's framework and state of the art; the concept of isoenergetic flow – an extended definition

This analytic study was done with the aim of improving and enriching the knowledge about the local physical phenomena encountered in fluid mechanics, elaborating a new physical & mathematical model of flow, by using a special type of coordinate systems - an intrinsic one. It represents a real deep insight into the still hidden theory of isoenergetic flow (some special topics in classical potential theory, fluid mechanics and aerothermodynamics).

This work is fully original – see the part of Section 10 (*Conclusions*), entitled "The original contribution of this work to the state of the art". It continues, extends and completes a series of works presented by this author at the 5th WCNA ([1]), at some WSEAS conferences ([2] – [5], [8]), and at a national conference in Romania ([6], [7]), having the aim of improving and enriching the knowledge about the local physical phenomena encountered in both fluid mechanics and magneto-fluid dynamics (extending the new physical & mathematical model of flow to magneto-gas dynamics, plasma included). A model of an isoenergetic flow of an inviscid fluid is introduced ([1], [2], [6]) 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. This model was extended to viscous Newtonian fluid flows ([1], [4], [6]), introducing some "zero-work" surfaces and a new physical quantity – Şelescu's "roto-viscous" **Ş** vector. The new first integrals (obtained by a procedure of eliminating the non-conservative terms in the respective equations) are similar to D. Bernoulli and D. Bernoulli–Lagrange ones. The PDE of the velocity potential, that of the isentropic surfaces, and those of Şelescu's vector lines and zero-work surfaces, are given.

The aim of the present work is to study and clarify the above phenomena for: 1. the vortex equation and its first integral (for a viscous incompressible fluid); 2. the motion equation and its first integral; 3. the continuity and flow rate equations; 4. the velocity potential equation, and 5. a possible case of first integrability for the system of equations (motion and continuity) for a steady flow (2, 3, 4 and 5 – for an inviscid compressible fluid). This work may be considered as the 3rd part of [1].

One calls static specific enthalpy (sensible or thermodynamic enthalpy) of a certain fluid small particle (considered as being a material point) the sum: $i = U + p/\rho = U + p\tau$, where: U is the specific internal energy of the respective particle; p is the local static pressure of the fluid flow; ρ is the local density of the fluid, and τ is the local specific volume of the fluid ($\tau = 1/\rho$). One still calls total (stagnation, impact) specific enthalpy (sensible or

*Tel.: +40 21 434 0083; fax: +40 21 434 0082
E-mail address: rselescu@aero.incas.ro; *URL:* http://www.incas.ro

A theoretical model to study the aerodynamic performances of horizontal axis windmills

Iosif Ţăposu*
S.C. INAV S.A., 44A, Ficusului Blvd. Sector 1, 013975 Bucharest, ROMANIA

Abstract: The paper describes a theoretical model to study the aerodynamic performances of horizontal axes windmills, more precisely a blade theory for this particular energy generator. Starting with the choose of reference systems are established formulas to determine the flow speed and the incidence angles distribution along the span, the aerodynamic forces and moments, the power of the blade as function of the basic parameters, also other quantities. For some particular cases it is performed a qualitative study, also established analytical formulas for computation of blade aerodynamic performances. It is proved that a high lift/drag ratio of airfoils along the span and a delayed stall have an important contributoin to reduce the start wind speed, also the high lift dolphin profiles seems to be an appropriate constructive solution to achieve this objective. The performances evolution with the time, respectively the dynamics of blade reduces to a second order nonlinear differential equation for which have to solve a Cauchy's problem. Finally we have to study a system of two nonliniar integral equations for which an appropriate and simple method of successive approximations is presented. A numerical example puts into evidence the quick convergence of the method and its performances, the theoretical instrument being useful in the design optimization process. Apart the selected airfoil sections it is shown that the geometrical torsion along the span is essential to reduce the start wind speed of energy generator that means an extention of upstream energy potential towards lower speeds. Three particular examples that correspond for power ranges very low, low and medium clarify more precisely these practical design aspects.
2010 Mathematics Subject Classification: 35 Partial differential equations; 65 Numerical analysis
Keywords: Horizontal axis windmill, blade, aerodynamics, dolphin profiles, power

1. Introduction

A very actual worldwide concern refers to the clean energy resources to ameliorate the conflict between economy development and the environment protection, mainly reduce the emissions. An alternative solution to solve this problem is the catch of kinetic energy that exists in air and/or water and convert it into useful energy, obviously this being a clean one. If refer to the available resources existing in air it is well known that these are spread non-uniformly around the world and so for the actual level of technology development the exploitation of this resource at a large scale cannot be profitable everywhere except those areas where have medium/high wind speeds many days per year. However, the resources existing in air and water are practically unexhausted and must remain under attention in parallel with continuously improvement of catch, convertion and storage technologies.

There are some well known technical solutions to catch the wind energy but the generators having horizontal axis is the most spread solution, the obtained powers varying from a few kW up to order of MW. An essential component of any wind turbine is the blade that transforms the kinetic energy existing in air into mechanical energy, respectively axial moment and any improvement of its catching ability will have positive consequences for the performances of whole machine. In spite of actual progress the author didn't identified in the literature a rigorous theoretical model to study the aerodynamic performances of the blade that may be used to perform optimization studies according to specific design criteria and the scope of this paper is to provide a such theoretical instrument.

From aerodynamic point of view it is an important difference between the blades that correspond to windmills having horizontal, respectively vertical axes due to the working principles. When have a horizontal axis then the upstream perturbation velocity is increasing approximatively linearly along the span, also the incidence angles haven't very high variations like as in the case of vertical axis. Choosing an appropriate torsion of the blade along the span then may realize an acceptable compromise regarding the requirement that in the design point the airfoil in every section along the span has a sufficiently high lift/drag ratio.

* Tel/Fax:+40 21 2323724, 2327667, 2326887
E-mail address: taposu@inav.ro; *URL*: http://www.inav.ro

INDEX

SCIENTIFIC COMMITTEE of the Conference

- **Dr. Mihai ARGHIR** – Université de Poitiers, France
- **Dr. Ştefan BALIN**T – West University of Timisoara
- **Dr. Ruxandra Mihaela BOTEZ** – École de technologie supérieure, Université de Quebec, Montreal, Canada
- **Dr. Mircea Dimitrie CAZACU**, UPB – University POLITEHNICA of Bucharest
- **Dr. Horia DUMITRESCU**, I.S.M.M.A. – Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy "Gheorghe Mihoc – Caius Iacob"
- **Dr. Horia ENE**, IMAR – Institute of Mathematics "Simion Stoilow" of the Romanian Academy, Bucharest
- **Dr. Constantin FETECĂU**, Technical University "Gheorghe Asachi", Iasi
- **Dr. Dorel HOMENTCOVSCH**I, Binghamton University, USA
- **Acad. Marius IOSIFESCU**, I.S.M.M.A. – Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy "Gheorghe Mihoc – Caius Iacob"
- **Dr. Mircea LUPU**, University "Transilvania", Brasov
- **Dr. Dan MATEESCU**, McGill University, Montreal, Canada
- **Dr. Alexandru MOREGA**, UPB – University POLITEHNICA of Bucharest
- **Dr. Cătălin NAE**, INCAS – National Institute for Aerospace Research "Elie Carafoli" (under the Aegis of the Romanian Academy), Bucharest
- **Dr. Adriana NASTASE**, RWTH, Aerodynamik des Fluges, Aachen, Germany
- **Dr. Titus PETRILA**, "Babeş-Bolyai" University, Cluj-Napoca
- **Dr. Dan POLIŞEVSCHI**, IMAR – Institute of Mathematics "Simion Stoilow" of the Romanian Academy, Bucharest
- **Dr. Ioan POP**, "Babeş-Bolyai" University, Cluj-Napoca
- **Dr. Mihai POPESCU**, I.S.M.M.A. – Institute of Mathematical Statistics and Applied Mathematics of the Romanian Academy "Gheorghe Mihoc – Caius Iacob"
- **Dr. Sorin RADNEF**, INCAS – National Institute for Aerospace Research "Elie Carafoli" (under the Aegis of the Romanian Academy), Bucharest
- **Dr. Ruxandra STAVRE**, IMAR – Institute of Mathematics "Simion Stoilow" of the Romanian Academy, Bucharest
- **Dr. Ion STROE**, UPB – University POLITEHNICA of Bucharest
- **Dr. Sanda ȚIGOIU**, University of Bucharest
- **Dr. Victor ȚIGOIU**, University of Bucharest

ORGANIZING COMMITTEE

Prof. Dr. Corneliu Bălan

E-mail: balan@hydrop.pub.ro

U.P.B. – "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

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

http://www.ima.ro/

Dr. Richard Şelescu

E-mail: rselescu@incas.ro

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

http://www.incas.ro

CONTENTS

ISSN 2067-4414
ISSN-L 2067-4414

ISSN National Center
Romanian National Library

BUCHAREST

2011