TRANSACTIONS
OF THE INSTITUTE OF AVIATION

Scientific Quarterly
5/2010 (207)

Aircraft Engines and Fuels
Aerodynamics and Mechanics of Flight
Materials Science

CONTENTS

M. Folusiak, Arkadiusz Kobiera, Piotr Wolański: ROTATING DETONATION ENGINE SIMULATIONS IN HOUSE CODE REFLOPS ............................................................................................... 3

G. Rarata, B. Naumienko: ALL ACTUAL NEEDS AND POSSIBILITIES TO PRODUCING BIO-JET FUEL ................................................................................................................... 24

P. Ruchała, A. Krzysiak: A METHODOLOGY OF UNSTEADY INVESTIGATIONS OF HELICOPTER AIRFOILS .......................................................................................................................... 32

Z. Skorupka: MAGNETOREOLOGICAL FLUIDS USE AS METHOD FOR ACTIVE CONTROLLING OF LANDING GEAR SHOCK ABSORBER CHARACTERISTIC ........................................ 48

Z. Skorupka, W. Kowalski, R. Kajka: TEST RIG PROTOTYPE FOR TESTING OF NOSE LANDING GEAR ELECTRICAL ACTUATOR DESIGNED FOR PASSENGER AIRCRAFT ........................................................................................................................ 57

J. Sznajder: DETERMINATION OF NON-LINEAR AERODYNAMIC CHARACTERISTICS OF AN AIRCRAFT USING A POTENTIAL FLOW MODEL AND VISCOUS AIRFOIL CHARACTERISTICS ........................................................................ 68

J. Sznajder, W. Stalewski: APPLICATION OF A PANEL METHOD WITH VISCOUS-INVISCID INTERACTION FOR THE DETERMINATION OF AERODYNAMIC CHARACTERISTICS OF CESAR BASELINE AIRCRAFT .................................................................................. 76

Edition is sponsored by Ministry of Science and Higher Education
The paper presents the results of three-dimensional preliminary simulations of a detonation propagating in Rotating Detonation Engine chamber. Simulations were performed using in-house code REFlOPS (Reactive Euler Flow Solver for Propulsion Systems)[1]. The description of the code and presented results are also included in MSc thesis of Folusiak and Swiderski [2]

1. INTRODUCTION

It is very difficult to perform the simulation of detonation process in the RDE due to physical and chemical nature of the detonation phenomenon and also due to geometrical complexity of the detonation wave structure in this engine. The first problem imposes the requirement of using proper numerical methods, which are able to reconstruct the discontinuities propagation processes in the flow. Due to problem specificity the simulation must include shock waves as well as rarefaction waves precisely and also separation areas, oblique waves and Mach stems must be taken into account. Thus, these simulations set high requirements for the solver. On the other hand as far as the RDE engine is concerned there is a strong coupling between phenomena at different time and space scales. The details of the wave's head are 10 μm order of magnitude or smaller, whereas these details strongly hang on large-scale chamber geometry, e.g. chamber diameter or passage gap. The coupling of different scales results also in a number of grid cells used to represent the simulation target. The number of cells is 10⁶ order of magnitude or many more.

2. DETONATION WAVE SIMULATIONS IN ANNULAR TUBE

The first simulations of the RDE engine were related to simple, annular chamber geometry. In order to simplify the computations there was assumed that the gas is stationary and the chamber is closed. The simulations performed in this case let one determine the structure of the detonation in macro scale (Fig 1, 2). The detonation propagating in annular tube is characterized by certain pressure, density and temperature gradients resulting from the occurrence of centrifugal forces as well as from wave front deformation.
Fig. 1. The shape of detonation wave front for hydrogen-air mixture
Pressure contours vs. time
Fig. 2. Density contours behind detonation wave front in annular tube

3. FLOW IN RDE CHAMBER
3.1. Chamber geometry. Boundary conditions

RDE chamber’s geometry was adapted from experimental research of Wolanski and Kindracki [3,4]. The computational grid of the considered domain’s cross section is shown on Figure 3.

Fig. 3. The computational domain and B.C.’s used in RDE engine simulation

The chamber dimensions are as follows (in mm’s)

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Value (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>External diameter</td>
<td>95</td>
</tr>
<tr>
<td>Internal diameter</td>
<td>75</td>
</tr>
<tr>
<td>Gap size</td>
<td>4</td>
</tr>
<tr>
<td>Total length</td>
<td>140</td>
</tr>
<tr>
<td>Working part length</td>
<td>80</td>
</tr>
</tbody>
</table>
The domain was split into 160 x 20 x 450 cells along $x$, $r$, $\phi$ directions respectively (Fig. 4), thus the resulting cell’s width is around 500µm along each direction.

Fig. 4. The computational grid used in RDE engine simulation
The REFLECTIVE boundary conditions have been set to the outermost radial boundaries of the domain. The PERIODIC boundary condition has been set on circumferential direction. On inlet, boundary condition of fixed values of velocity, pressure and temperature was set, so that total pressure and temperature are to be constant parameters of feeding system. The following flow parameters have been assigned to the boundary:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Feeding pressure</td>
<td>11 bar</td>
</tr>
<tr>
<td>Feeding temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Feeding velocity</td>
<td>76.2 m/s</td>
</tr>
<tr>
<td>Mixture</td>
<td>hydrogen - air</td>
</tr>
<tr>
<td>Stoichiometry coefficient</td>
<td>1.0</td>
</tr>
</tbody>
</table>

PRESSURE OUTLET boundary condition has been set to the other outermost axial boundary. The subsonic flow is relaxing at specified length to the conditions defined by user. Otherwise, if the flow is supersonic, TRANSMISSIVE boundary condition is set to the considered boundary (which is equivalent to infinite length of relaxation distance).

The flow in steady state without detonation

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Feeding pressure</td>
<td>0.5 bar</td>
</tr>
<tr>
<td>Feeding temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Mixture</td>
<td>hydrogen - air</td>
</tr>
<tr>
<td>Stoichiometry coefficient</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Before any analyses of reactive flow were performed, the steady state, flow has been simulated. Because of the fact that the chamber as well as the flow are axisymmetric, two-dimensional computations have been performed.

Then the solution was extrapolated into three-dimensional case (along φ direction). The results are presented on Figures 5 and 6. It may be said that solution is converged after 1000 µs.

The solution has been taken for further analysis as initial condition for the initiation of detonation.

Fig. 5. Stationary flowfield in RDE chamber (without detonation). Two-dimensional case
**Mach number**

**Pressure**
Temperature

Fig. 6. Stationary flowfield in RDE chamber (without detonation). Three-dimensional case

4. DETONATION WAVE PROPAGATION IN RDE CHAMBER

The last phase of simulations was performing the computations of the detonation wave propagation through the cylindrical domain. The previous results of stationary flowfield have been used as the initial conditions for this simulation. The mixture is ignited in such a way that only one detonation wave is propagating in circumferential direction.

The other is an opposite shockwave, which is propagating through the half of the domain which is initially out of the range of chemistry. Reaction mechanism contains only one reversible reaction, which is defined as follows:

$$2\text{H}_2 + \text{O}_2 = \text{H}_2\text{O} + \text{H}_2\text{O} \ 5.5\times10^{16} \ 0.0 \ 8400.0$$

Previous simulations showed, that 1-step reaction mechanisms as the one above is not appropriate for detonation modeling. Detonation transits in this case into the stationary deflagration near the annular gap. All the fresh mixture that should refill the combustion chamber is being burned there immediately. This is caused by the numerical diffusion, high width of the grid and simplicity of the reaction mechanism.

However, the artificial deflagration may be avoided. It is typical for such simple chemistry models, that chemical reactions are artificially limited by the pressure in the current cell. In these simulations, chemical reactions are considered only if the pressure of 20 bars is exceeded in the computational volume. This approach was successfully used by Davidenko et al [5].

Mixture creation process cannot be modeled in the code yet. Thus, it was decided to inject premixed stoichiometric mixture of reactants directly to the collector. Because of high pressure, flow is choked in the annular gap, and mass flux is limited by its geometry. In the Rotating Detonation Engine, only fuel flows through the bottleneck, and oxidizer is injected from the outer boundary just behind the gap. Because of the lack of an appropriate mechanism of mixture creation, it has been assumed that stoichiometric and homogeneous mixture is created at a distance of 2.5 mm from the gap end.

Therefore, combustion zone has been limited to the region 4 cells below the gap. Ignition process is showed on Figures 7 and 8.
Time = 0 microsec.
p_{\text{max}} = 17.8 \text{ bar}

Time = 5 microsec.
p_{\text{max}} = 37.8 \text{ bar}
Fig. 7. Ignition process used in RDE's simulation. Pressure contours
Time = 0 microsec.
T_max = 1400 K

Time = 5 microsec.
T_max = 3470 K
Fig. 8. Ignition process used in RDE's simulation. Temperature contours
After ignition the rotating detonation develops and propagates along the channel.
The first analyses of the results showed that the structure of the detonation front is three-dimensional and highly influenced by the structure of the initial supersonic flow of the mixture.
The structure of the wave is completely different in the inner and outer surface of the chamber (Fig. 9, 10).
Time = 200 microsec.  
\( p_{\text{max}} = 47.5 \text{ bar} \)

Time = 210 microsec.  
\( p_{\text{max}} = 42.0 \text{ bar} \)
Fig. 9. Propagation of the detonation in RDE. Pressure field
Time = 190 microsec.

T_{max} = 3140 K

Time = 200 microsec.

T_{max} = 3280 K
Fig. 10. Propagation of the detonation in RDE. Temperature field
For better understanding of the phenomena occurring in the flow, the two dimensional structure of the wave has been also studied. The structure of the wave at the inner wall is presented on Figure 11.

Figure 11. Structure of the waves at the inner wall

On the picture presented above, one can see Mach stem wave (MS), resulting from the collision of two waves in A-point, because of the fact that dynamic equilibrium must be conserved in this point. Also the chemical influence – chemistry limiting line, which plays important role in this process, must be pointed out. In more complex simulations, the shape of MS wave will depend on the injection method and the mixing process.

A-point is the triple point in this case. However, it is not the same triple point like in cellular structure of detonation, which cannot be observed in macroscale simulation. LD is the leading detonation wave or detonation head. At the right end of LD wave, one can see another wave, between this point and long tail wave (LT). This short wave propagates downstream. There is also one more wave which propagates upstream – transverse shock (TS). The shape of the shock wave, as an iso-surface of the pressure, is shown on Figure 12.
Summary

Performed computations of the detonation wave propagation in the RDE engine demonstrated that the REFloPS code is a useful tool in research of this type of engine. The preliminary results for the three-dimensional simulation of the RDE has given some information about the structure of the detonation wave. It also has risen some question e.g. about the role of deflagrative combustion in the region of mixing of the hot gases with the fresh mixture. The simulations showed also that special attention must be paid the interactions of the numerical diffusivity and proper mechanism of the chemical reactions.

REFERENCES

SYMULACJE SILNIKA Z WIRUJĄCĄ DETONACJĄ (RDE) W KODZIE REFLOPS
Michał Folusiak

**Abstrakt**

W artykule przedstawiono wyniki trójwymiarowych symulacji detonacji w komorze silnika z wirującą detonacją (RDE). Symulacje przeprowadzono przy użyciu kodu REFloPS, który jest wynikiem pracy magisterskiej dwóch pracowników Instytutu Lotnictwa.
ACTUAL NEEDS AND POSSIBILITIES TO PRODUCING BIO-JET FUEL

Bohdan Naumienko, Grzegorz Rarata
Institute of Aviation

Summary

Since the beginning of the century, various studies, reports and popular scientific papers have investigated the potential for use of biofuels in aviation. In its recent actions EU also perceives the need for the introduction of sustainable biofuels to help reduce dependence on fossil fuels in air transport and reduce GHG emissions by the air industry. Thus, various feedstocks and conversion technologies for production of biofuels for aviation are currently being investigated within EU (SWAFEA, ALFA-BIRD and Clean Sky JTI). The current EU policy states that sustainable bio-jet fuels are the only option that can be delivered over the medium term, in time to make a significant contribution to EU 2050 emission reduction targets [11].

However, it seems that commercial producers of fuels cannot justify investment in a Bio-Jet fuel production facility until guaranteed availability of sustainable feedstock and clear legislation. EU government is currently taking necessary steps to open a long-term option for the aviation biofuel production in the local market and to support its development. Whereas microalgae is certainly a feedstock considered by many future Bio-Jet producers. Only microalgae seems to offer vast potential as a source of aviation biofuels, however on the more distant horizons of an industry, in which long-term time scales are certainty. Besides, microalgae co-products are utilised at making complex organic compounds like B and C vitamins and beta-carotene that are used as fragrances, flavourings, pigments and supplements. Generally, the research done so far have demonstrated that only a portion of the crude algal oil is suitable for making biodiesel fuel, and most of it can be used to produce gasoline and Bio-Jet like fuels.

If we look from the space at Earth in the night, the view of electric lights concentration does not leave any doubt: we live in the world with big and increasing demand for energy.

In European Union which the member is Poland, the use of energy and GHG emission is not dramatic, but still needs some reflection. If we approximate the trend from year 2005, then in year 2030 the energy consumption increases 11% and CO₂ 5.4%.

This situation forced the European Commission (EC) to meet the following goals until year 2020, first time as binding for all EU members:
- 20% decrease in GHG emission
- 20% increase in renewable energy sources
- 20% increase in energy efficiency
- 10% increase in biofuel use
Additionally to the problems with increasing demand for energy, EC also sees a matter of supply reliability. After Russia-Ukraine gas dispute in January 2009, it turns out that mitigating of energy crisis depends as usual on source differentiation and as well on monitoring of current energy market, and the Energy Commissioner Andris Piebalgs appealed to European Parliament for complying law regulations with new threat. In meantime, EC itself issued the proposal of a new regulation adopting the old one No.736/96 which should in principle allow monitoring of energy market and investment projects at EU scale.

In the context of these problems, the Polish objective that during the following next 20 years we should provide 15% of energy needs basing one renewable sources (RS), it seems to be hard to achieve. But maybe we can get that 10% of biofuel.

Tab. 1. Polish potential of renewable energy sources (Wiśniewski G., 2006)

<table>
<thead>
<tr>
<th>Source of energy</th>
<th>Percent of national technical infrastructure</th>
<th>Percent of national use</th>
</tr>
</thead>
<tbody>
<tr>
<td>Biomass</td>
<td>43.1</td>
<td>94.8%</td>
</tr>
<tr>
<td>Sun</td>
<td>25.4</td>
<td>0.12%</td>
</tr>
<tr>
<td>Wind</td>
<td>16.1</td>
<td>0.17%</td>
</tr>
<tr>
<td>Geothermal</td>
<td>12.6</td>
<td>0.29%</td>
</tr>
<tr>
<td>Water</td>
<td>2.8</td>
<td>4.62%</td>
</tr>
</tbody>
</table>

Let us see what is the Polish potential of renewable energy sources (Tab. 1). The table above show clearly that the first resource and its technical potential is biomass. Therefore, from SE² reasons (security, ecology, economy) Polish the main interest should be getting started the clean transport industry from production of biofuel with use biomass and the sun energy.

1. CURRENT CLASSIFICATION OF BIOFUELS

There seems to be lack of a clear classification for biofuels. The most important feature of all biofuels is that they are derived from biomass. That means to produce them, sources of carbon, hydrogen, oxygen, nutrients, and sunlight are required. Basically for making biofuels we need glucose, cellulose or oil.

Oil producing plants, like those used in vegetable oils, or algae, can be used much like fossil sources of oil. Oil derived from such sources of biomass is suitable raw material for creating simple biodiesel that can be burned by cars or further processing to synthetic biofuel kerosene like (replacement for petrochemical Jet A or Jet A-1). The latter type of biofuel is the effect of recent technological innovations that have created the fields of advanced bio-fuels, which focus on non-food sources and drinking water sources. So, these kind of biofuels prevents the debate on whether growing crops for fuel will result in fewer available food crops. This new forms of fuels can literally be called green, as they are derived from green algae. Some algae, especially microalgae, species characterize very high oil content. The oil may be processed like other oil from crops, for example via hydrotreatment or other chemical reaction processes that use catalysts.

Many developed countries, with USA ahead, are now doing extensive research on algae. Some of them are easy to cultivate and grow extremely quickly, compared to the traditional oil crops. According to some estimates, some algae can produce tens times as much oil as one acre of rape and may utilise municipal sewage as a source of nutrients.
As a matter of fact, there are still many doubts about the future use of biofuels, not only in Poland. A good understanding and strong backing of bioenergy by the wider European public seems to be an essential background for policies supporting the introduction and wider use of biofuels. Thus, a better understanding of the acceptance and public perception of biofuels and resulting strategies to gain higher public support have to be a question of great importance.

Picture below (Fig. 1) presents different kinds of renewable fuels together with the sources of biomass and conversion technologies for their production.

Peering into the future seldom produces a clear picture but nowadays there is a key question about future consumption of biofuels. Currently most research into efficient algal-oil production is being done in the private sector, but predictions from small scale production experiments prove that using algae to produce biodiesel may be the only viable method by which to produce enough automotive and aviation fuel to replace current world usage. Another fact is that the profitability of biofuels production is extremely variable. Due to the volatile price nature of oils, its major feedstocks, biofuels profitability can change rapidly from month to month and member countries of EU, depending also on changeable crude oil prices and legislation.

In fact, the rise in oil prices is the most important factor increasing the competitiveness of alternative fuels, including biofuels. The unprecedented recent rise in oil prices has prolonged opportunities for energy conservation and generated increased supply from alternative fuel sources. While these adjustments may eventually decrease oil prices, most forecasts do not show real prices falling below 50 dollars per barrel. Emerging new technologies of III generation biofuels from microalgae provide a clean, renewable form of energy that would help Poland to find its energy independence. Thus, the most obvious finding so far is that new technologies resulting from research and development are the key to developing a sustainable biofuel industry that meets national targets. These technologies include enhanced production systems (sustainable management tools; better data, models, and decision tools) as well as the integration of feedstock
production with conversion and utilisation.

Currently, it is obvious that biofuels from algae could be a meaningful part of the solution in the future, especially taking into account the EU requirements towards year 2020 (that biofuels will account for 10% of European fuel market). These assumption applies also to the aviation industry that will need carbon-neutral biofuels as a feasible way to reduce its reliance on fossil fuels and cut its greenhouse gas emissions. In Europe there have been lack advanced research in this field, compared to the USA.

One reason for this situation may be fact that there is a significant delay in the reliance on advanced alternative fuels in Europe in general (Tab. 2).

**Tab. 2. World leaders in biofuel production [2]**

<table>
<thead>
<tr>
<th>Country</th>
<th>Production (kton, year 2007)</th>
<th>Contribution (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>USA</td>
<td>13 793</td>
<td>40</td>
</tr>
<tr>
<td>Brazil</td>
<td>11 397</td>
<td>33</td>
</tr>
<tr>
<td>Germany</td>
<td>2 779</td>
<td>8</td>
</tr>
<tr>
<td>China</td>
<td>1 202</td>
<td>3</td>
</tr>
</tbody>
</table>

Bio-derived Synthetic Paraffinic Kerosene (Bio-SPK), made from Jatropha, Camelina, algae or halophyte feedstocks, is so far the most promising candidate in the USA for alternative jet fuel and test flights have successful proven its feasibility as a replacement for conventional jet fuel [3].

2. AVIATION PETROCHEMICAL FUEL (JET)

Today the most commonly used fuels for commercial aviation are petrochemical Jet A and Jet A-1 which are produced to a standardized international specification. There are kerosene grade fuels produced by the most oil refineries around the worlds and are suitable for most turbine engine aircrafts. Only a fraction of this production, in South Africa, comes from coal conversion via CTL process (Coal to Liquids) [4].

Aviation fuels such as the most common aviation fuels utilized by the commercial airline industry – Jet A or Jet A-1 – must satisfy rigorous specifications to operate successfully in flight, including physical properties as well as “fit-for-purpose” operating criteria. Aircraft, for example, operate at temperatures ranging from greater than 55 °C at ground level to less than -60 °C at altitude, and at elevations ranging from ground level up to 12 km. In addition, turbine engine manufacturers are increasingly utilizing the fuel prior to combustion as a hydraulic fluid for actuating advanced engine features. This extraordinarily broad range of operating conditions and functions requires special examination of alternative jet fuel cold flow (viscosity), flash point, distillation, energy density properties, as well as engine system compatibility.

Thus, Certification of jet fuel is a complex, expensive and lengthy process that requires clear technical standards, international cooperation and industry support. The US standards for all fuels are set by the ASTM International organization.
ASTM has recently established a new standard (ASTM D 7566-09) for Aviation Turbine Fuel Containing Synthesized Hydrocarbons [5].

Tab. 3. Estimated oil production from different biomass sources (authors’ comparison)

<table>
<thead>
<tr>
<th>Biomass</th>
<th>Oil content (liters/hectare)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Corn</td>
<td>150</td>
</tr>
<tr>
<td>Soybeans</td>
<td>400-500</td>
</tr>
<tr>
<td>Sunflower</td>
<td>800-1000</td>
</tr>
<tr>
<td>Rapeseed</td>
<td>1000-1500</td>
</tr>
<tr>
<td>Oil Palm</td>
<td>2400-2500</td>
</tr>
<tr>
<td>Microalgae</td>
<td>20000-57000</td>
</tr>
</tbody>
</table>

It is essential that certification policy be technology neutral, requiring physical properties and fit-for-purpose specifications assessed only on final fuel blends as used and not the blending components.

3. BIOMASS DERIVED JET FUEL

There are current research analyses that microalgae are potentially the most promising feedstock for producing large quantities of sustainable aviation biofuel Jet A like. The lipid and fatty acid contents of microalgae vary in accordance with culture conditions and may be as high as 70 to 85% on a dry weight basis [6]. Such high lipid contents, exceed that of most terrestrial oil crops, thus traditional oilseed crops are not the most productive or efficient source of vegetable oil. Micro-algae is, by a factor of 8 to 25 for palm oil, and a factor of 40 to 120 for rapeseed – the highest potential energy yield terrestrial vegetable oil crops (Tab. 3). Additionally, microalgae present multiple possibilities for fuels production – biodiesel, ethanol, methane, jet fuel, biocrude and more – via a wide range of process routes. Each of these process routes presents its own set of opportunities, parameters, dynamics and challenges. Besides, microalgae can be grown quickly in salt water (e.g. in the desert).

Solazyme was one of the first companies in the world that started producing a variety of renewable products from algae oil, including renewable diesel that meets ASTM D975 standard and renewable jet fuel that meets all 11 key tested criteria for ASTM D1655 (the world’s first algae-based jet fuel) – the latter one in 2008 [7]. Solazyme’s technology is feedstock non-dependent, which allows commercial scale production plants to be placed worldwide, adjacent to many non-food biomass sources and waste streams (Fig. 2).
Another private algae fuel company is Sapphire Energy, which has raised more than $100 million from Bill Gates’ investment firm Cascade Investment, as well as ARCH Venture Partners, Wellcome Trust and Venrock. Sapphire announced that it will make 1 million gallons of algae-based diesel and jet fuel per year by 2011 and 100 million gallons per year by 2018 [8].

UOP, a Honeywell company, gained contract from DARPA in 2007, for producing renewable jet fuel for military (Synthetic Paraffinic Kerosene, Bio-SPK). UOP LLC, a Honeywell company is a leading international supplier and licensor of process technology, catalyst, adsorbents, equipment and consulting services to petroleum refining, petrochemical, and gas processing industries. UOP technology for the production of clean, high quality fuels and petrochemicals is used today in almost every refinery around the world.

The UOP proprietary process is based on the hydroprocessing technology that has been used in refineries for almost 50 years. Hydrogen is used to remove oxygen from the natural oil and then the product is further isomerized to get to the product properties needed (Fig. 3,4).
Developed using plants such as jatropha, algae and camelina, bio-SPK fuel has been successfully utilized by commercial aircrafts at a blend ratio of up to 50 per cent with traditional jet fuel. Currently tests are going on and various SPK fuels are now waiting for the regulatory approval for use on commercial flights. Additionally, a coalition of companies from across the aviation industry, including Boeing, Continental Airlines, Air New Zealand and Rolls-Royce, have also found that Bio-SPK outperforms traditional petroleum-based aviation fuels.

A study revealed that Bio-SPK delivers a cleaner burn resulting in improved fuel efficiency and less wear on engine components [9].

4. Technological Background

The renewable jet process compiled by UOP LLC company is a good example of novel technological methods that may be implemented in aviation biofuels production. The process can convert a variety of refined natural oils and fats including edible and non-edible natural oils, tallow and algal oils. The renewable jet process uses a selective cracking step which reduces the natural oil C16-C18 carbon chain lengths to carbon chain lengths in the C10 to C14 range for jet fuel. The renewable jet process is based on UOP’s Ecofining™ process, which is commercially available for the production of green diesel produced from biofeedstocks. While the Ecofining unit can produce up to 15% of Bio-SPK jet fuel, as a co-product with diesel, this new process is designed to maximize the yield of Bio-SPK to 50-70% [10]. This is achieved by optimizing the catalytic processes of deoxygenation, isomerisation and selective cracking of the hydrocarbons present in natural oils and fats to yield a high quality, ultra-low sulphur jet fuel that meets Jet A-1 specifications, including freeze point of -47°C and flash point of 38°C (Fig. 5). Co-products from this new process are diesel and naphtha range material. The process can be adjusted to produce a specific freeze point of the Bio-SPK or can alternately be operated in a diesel mode.
LITERATURE


Streszczenie

W artykule przedstawiono skrót aktualnych osiągnięć technologicznych na temat potencjalnego zastosowania biopaliw w lotnictwie. Zaznaczono również, że ostanie dyrektywy Komisji Europejskiej także faworyzują wprowadzanie biopaliw do transportu lotniczego, celem zmniejszania zależności od importowanych paliw ropopochodnych oraz obniżania emisji gazów cieplarnianych. Wymieniono różne podejścia technologiczne do zagadnienia produkcji biopaliw dla lotnictwa.

A METHODOLOGY OF UNSTEADY INVESTIGATIONS OF HELICOPTER AIRFOILS

Andrzej Krzysiak, Paweł Ruchała
Institute of Aviation

Summary
The paper describes a methodology of unsteady experimental investigations of helicopter airfoils. It has been implemented during tests of helicopter airfoil, carried out in Aerodynamics Department of Institute of Aviation for PZL-Świdnik and Ministry of Science and Higher Education (MNiSW – Ministerstwo Nauki i Szkolnictwa Wyższego) as a part of grant: “Development and deployment of new generation of design, technological and material solutions for main rotor and airframe elements of PZL W-3A Sokół helicopter”.

The tests have aimed to modeling (in the wind tunnel) the dynamic stall phenomena incidence, which may appear on main rotor blades during forward flight. It causes a strong vibration of blades, thus defining a considerable limit of helicopters’ performance.

The dynamic stall phenomena is caused by fast angle of attack transition, which appears during forward flight. A similar transition on tested model was evoked by its oscillations with requested amplitude and frequency. The mechanism causing the oscillations of model and the measurement equipment have been described further.

The discussed methodology covers pressure distribution measurements, basing on measurement of local static pressure on the surface of respectively adapted model. Because the measurements of pressure are not simultaneous, the pressure coefficient distribution (as a function of time and angle of attack) has been approximated using Fourier series. The coefficients of lift and pitching moment have been calculated as a result of integration the pressure coefficient distribution. An algorithm of calculation has been described also.

Acknowledgements
The investigation has been carried out as a part of the grant: “Development and deployment of new generation of design, technological and material solutions for main rotor and airframe elements of PZL W-3A Sokół helicopter” (“Opracowanie i wdrożenie nowej generacji rozwiązań konstrukcyjnych, technologicznych i materiałowych dla wirnika nośnego i elementów płatowca śmigłowca PZL W-3A Sokół”) for Ministry of Science and Higher Education (Ministerstwo Nauki i Szkolnictwa Wyższego) and PZL – Świdnik.
Notation:

\( a_i \) – \( i=0...N \) the coefficients of function approximating \( C_p(t) \) and \( \alpha(t) \) function

\( b_i \) – \( i=1...N \) the coefficients of function approximating \( C_p(t) \) and \( \alpha(t) \) function

\( c \) – airfoil chord [mm]

\( C_0 \) – the coefficient of ESP-16HD sensor’s characteristic [psi]

\( C_1 \) – the coefficient of ESP-16HD sensor’s characteristic [psi/mV]

\( C_2 \) – the coefficient of ESP-16HD sensor’s characteristic [psi/mV^2]

\( C_3 \) – the coefficient of ESP-16HD sensor’s characteristic [psi/mV^3]

\( C_4 \) – the coefficient of ESP-16HD sensor’s characteristic [psi/mV^4]

\( C_M \) – pitching moment coefficient (about quarter-chord point) [-]

\( C_{M_{\text{min}}} \) – minimal pitching moment coefficient (about quarter-chord point) [-]

\( C_N \) – normal force coefficient [-]

\( C_p \) – pressure coefficient [-]

\( C_T \) – tangential force coefficient [-]

\( C_L \) – lift coefficient [-]

\( C_{L_{\text{max}}} \) – maximal lift coefficient [-]

\( D \) – index of angular location sensor division [-]

\( D_0 \) – index of angular location sensor division related to \( \alpha=0 \) [-]

\( f \) – frequency of oscillation [Hz]

\( Ma \) – Mach number [-]

\( N \) – number of components of Fourier series approximating \( C_p(t) \) and \( \alpha(t) \) function [-]

\( p \) – local static pressure [kPa]

\( p_0 \) – total pressure of freestream [kPa]

\( p_S \) – static pressure of freestream [kPa]

\( q \) – dynamic pressure of freestream [kPa]

\( R' \) – radial coordinate of blade airfoil, normalized by blade radius [-]

\( Re \) – Reynolds number [-]

\( t \) – time [ms]

\( T \) – air temperature [K]

\( U \) – voltage registered by ESP-16HD sensor [mV]

\( U_T \) – voltage registered by temperature channel of ESP-16HD sensor [mV]
1. INTRODUCTION

Fast, periodic transitions of angle of attack appear very often on helicopter main rotor blades. They are caused by i.a. asymmetry of flow around rotor during forward flight. A transition of angle of attack, as a function of angle of blade rotation $\Psi$, is similar to sinusoid (in predominant part of rotor blade, especially in external part – for $R^*>50\%$). It has been shown in fig. 1.

Fig. 1: A dependency of blade’s angle of attack and angle of blade rotation, in comparison with the sinusoid ([3])

Because of fast transitions of blade’s angle of attack, in some conditions there appears the so called “dynamic stall” phenomena. The range of its incidence has been shown in Fig. 2.
The dynamic stall means, that flow separation on blade’s upper surface has limited range, even after exceeding a critical (for static conditions) angle of attack. Because of it, much higher maximal lift coefficient than in static conditions can be achieved. The $dC_L/d\alpha$ derivate increases also. This effect is connected with a vortex created on blade’s leading edge after exceeding a critical (for static conditions) angle of attack. The vortex moves toward trailing edge during angle of attack’s increase, causing a steep decrease of pitching moment.

Fig. 2: The incidence of dynamic stall phenomena ([3])
The moment the vortex leaves blade’s trailing edge, a lift force steeply decreases. Because the dynamic stall phenomena lasts very shortly, it may cause a rotor blade’s vibrations, which constitute an important limitation of helicopters’ performances [3].

2. TEST TECHNIQUE
2.1 N-3 Wind Tunnel

The dynamic stall phenomena, described above, has been sifted during investigations of helicopter main rotor airfoil. The investigation has been carried out on the model which was performing an oscillations with requested amplitude and frequency.

During the investigation a N-3 high-speed wind tunnel (shown in Fig. 3) has been used. It’s a blow-down wind tunnel with partial re-circulation of the flow, which enables investigations in the subsonic, transonic and supersonic flow regimes. Available Mach number: \( Ma = 0.2 \div 1.2 \) (with continuous control of flow velocity), \( Ma = 1.5 \) or \( Ma = 2.3 \). Only when respective nozzle is mounted \( Ma = 1.5 \) and \( Ma = 2.3 \) are available.

![Fig. 3: A part of N-3 high-speed wind tunnel (photo by A. Dziubiński, IoA)](image)

N-3 wind tunnel has got a close test chamber, with 0.6x0.6 m square cross-section. Each side wall of test section is equipped with two double windows of 0.25 m diameter. Top and bottom walls are disposable. Solid and perforated walls may be mounted. During described tests the perforated walls have been used to decrease the walls’ interference in the flow.

An exact description of N-3 wind tunnel and airfoils’ test technique have been published in the paper [10]
2.2 TESTED MODEL

An investigated airfoil model is a rectangular wing, without aerodynamic and geometric twist. Its chord length equals 0.2 m and wing span –0.6 m. The wing span equals test chamber’s width, so two-dimensional flow has been assumed (test chamber walls works as the endplates). The model, mounted in a wind tunnel test chamber, is shown in Fig. 4.

![A model of investigated airfoil mounted in a wind tunnel test chamber (photo by S. Podgródny, IoA)](image)

The tested model is all-metal and hollowed. Its upper surface part may be disassembled. It enabled mounting a pressure sensors inside the model, as shown in Fig. 5. Three 16-channel ESP-16HD by Pressure System Inc. pressure sensors have been used. The measured range equals ± 5 psi (for one of sensors, no. 59) or ± 10 psi (for two extant sensors, no. 60 and 61). Sensors’ inputs have been connected using thin pipes with appropriate pressure measurement points – i.e. 0.5 mm holes drilled on the model’s surface. 69 holes have been drilled. Unfortunately, not all of them have been used, because only 48 sensors’ channels were available. Arrangement of pressure measurement points is sketched in Fig. 6.
The airfoil's flow conditions were simulated in the wind tunnel by its oscillations around the so called „base angle” $\alpha_B$. Thus the angle of attack is a sum of base angle and instantaneous value of oscillation angle:

$$\alpha \approx \alpha_B + \Lambda \alpha \cdot \sin(2\pi \cdot f \cdot t)$$

Base angle was regulated by angle of attack mechanism, used in static investigations. This mechanism was joined with rod mechanism generating oscillations, which has been powered by electric engine. The whole device, shown in Fig. 7, has been connected with one side of the model's rotation axis. On the other side the angular location sensor ROC-412 by Heidenhain, registering the instantaneous value of angle of attack, has been mounted.
A device described above enables transition of oscillation frequency (up to 10 Hz) through the change of engine’s angular velocity. Setting the oscillation amplitude (+/- 5° or +/- 10°) was also possible by changing the point of rod’s fixing to engine’s fly-wheel. The whole device (with the angular location sensor) has been used also as a model’s fixing in the test chamber – the mechanism and the angular location sensor have both been mounted in the windows in the side walls of the test chamber.

3. ALGORITHM OF CALCULATION PROCESS

The investigation results have been obtained in a four-stage process. The stages are as follows:
• Registration of measured values (during the test)
• Processing of every made run (test cycle), i.e. calculation of lift and pitching moment coefficients
• Analysis of results’ correction
• Conclusions

For data registration (stage 1) the DYNAK program has been used. It handles the data acquisition devices connected with used sensors. The DYNAK program is a part of SPITA N-3/PC measurement system.
Parameters measured during the test are, among others:

- instantaneous value of static pressure on model’s surface
- instantaneous value of angle of attack
- static and total pressure of freestream
- freestream temperature

The parameters listed above are saved by DYNAK program in various (both text and binary) files, which are a source for PdN-3 program, aimed at processing the runs’ results.

PdN-3 program’s calculations results are saved as a ASCII file. The program can make the graphs on its own or export data for other software – Grapher for example.

3.1. REGISTERED DATA

Usually one run contains several measurements for various base angles. During every measurement 256 data sets have been registered. Every set contains the information on instantaneous value of angle of attack (measured by ROC-412 angular location sensor) and on static pressure instantaneous values (measured by ESP-16HD pressure sensors).

In effect 256 instantaneous values of angle of attack and static pressure (for every measurement point) are available in whole measurement. It’s not enough, however, for pressure distribution’s calculating. Every instantaneous value is registered in different moment, because of measurement’s duration.

Because of angle of attack’s continuous transition, every instantaneous value of pressure corresponds with different angle of attack’s value.

A sequence of data registration is shown in Fig. 8.

![Fig. 8: A sequence of data registration](image_url)
Except for data sets described above, during every run the following parameters of freestream have been registered: temperature, static pressure and total pressure. Temperature of every ESP-16HD sensor has been measured also.

### 3.2. PROCESSING

During processing stage the following actions have been taken:

1. Obtaining instantaneous value of angle of attack
2. Obtaining instantaneous value of static pressure on the model’s surface
3. Obtaining instantaneous value of pressure coefficients $C_p$
4. Interpolation of $C_p(t)$ and $\alpha(t)$ – obtaining a pressure coefficient distribution for requested time
5. Integrating of the pressure coefficient distribution $C_p(x)$ – obtaining lift and pitching moment coefficient
6. Obtaining of Reynolds number

Because of lack of algorithm for interference correction during unsteady investigations, no corrections have been implemented.

Individual operations have been shortly described further:

#### 3.2.1 CALCULATION OF INSTANTANEOUS VALUE OF ANGLE OF ATTACK

The ROC-412 angular location sensor is equipped with code-disc with 4095 divisions. Thus the information about angle of attack’s value is an index of division, saved in a binary file. Basing on it, when number of division related to $\alpha=0$ is known, angle of attack may be calculated using equation:

$$\alpha = \alpha_B - \left(D - D_0\right) \cdot \frac{360}{4095} \text{[deg]}$$

where:

- $\alpha_B$ – base angle
- $D$ – number of division registered during the test
- $D_0$ – number of division related to $\alpha=0$

Direction of positive angle of attack is opposite to the direction of division index’s increase. Therefore in the above equation appears the "minus" sign.

#### 3.2.2 CALCULATION OF INSTANTANEOUS VALUE OF STATIC PRESSURE ON MODEL’S SURFACE

Instantaneous value of static pressure have been calculated in 2 stages:

1. Calculation of voltage of respective ESP-16HD sensor channel, basing on binary data registered during the test
2. Calculation of pressure, basing on voltage obtained as above and on the channel characteristics:

$$p = C_0(U_i) + U \cdot C_1(U_i) + U^2 \cdot C_2(U_i) + U^3 \cdot C_3(U_i) + U^4 \cdot C_4(U_i)$$

$C_0...C4$ coefficients are different for each channel, therefore it was needed to obtain 48 coefficients sets. Channel characteristics depend also on sensor’s temperature, which is related to voltage of respective ESP-16HD sensor’s temperature channel.

A dependency between temperature channel’s voltage and $C_0...C4$ coefficients has been obtained basing on producer’s information. The results of calibrations made by Aerodynamic Division of Institute of Aviation have been used as well.
3.2.3 CALCULATION OF INSTANTANEOUS VALUE OF PRESSURE COEFFICIENT $C_p$

Instantaneous values of pressure coefficient have been obtained using equation:

$$C_p = \frac{p - p_s}{q}$$

where:

- $p$ – static pressure on a model’s surface (in a respective measurement point)
- $p_s$ – static pressure of freestream (measured by Solartron sensors, which are a part of SPITA N-3/PC system)
- $q$ – dynamic pressure of freestream, obtained with equation:

$$q = 0.7 \cdot p_s \cdot Ma^2$$

- $Ma$ – Mach number, obtained using equation:

$$Ma = \sqrt{\frac{2}{5} \left( \frac{p_s}{p_0} \right)^{3/2}} - 1$$

- $p_0$ – total pressure of freestream (measured by Solartron sensors)

It is assumed that static, dynamic and total pressure of freestream and its Mach number are constant during the whole measurement.

3.2.4 AN INTERPOLATION OF $C_p(t)$ AND $\alpha(t)$ DEPENDENCIES

Basing on 256 instantaneous values of pressure coefficient $C_p$ (for each measurement point) a $C_p(t)$ function has been approximated as a Fourier series:

$$C_p(t) = a_0 + \sum_{n=1}^{N} \left( a_n \cos(n \omega t) + b_n \sin(n \omega t) \right)$$

where:

- $\omega$ – frequency of model’s oscillation
- $a_0, a_1, ..., a_N$ and $b_1, b_2, ..., b_N$ – coefficients of Fourier series, obtained with least squares method.

A dependency shown above enables calculation of instantaneous values of pressure coefficient $C_p$ for requested time (identical for all measurement points). In the effect, pressure coefficient distribution, related to the requested time, may be obtained.

The instantaneous angle of attack values have been calculated in the same way.

It is clear, that $C_p(t)$ functions have been approximated as periodic functions, which period equals a period of model’s oscillations. It’s equivalent to the assumption that transitions of pressure distribution are the same during every period of oscillations.

This assumption is quite well fitted in practice. It has been shown in Fig. 9, which illustrates a dependency between pressure coefficient (in one of measurement points) and time, during the next periods of oscillation. The values obtained with Fourier series have been shown also.
3.2.5 INTEGRATING OF PRESSURE COEFFICIENT DISTRIBUTION

An essence of pressure distribution measurement is calculation of aerodynamic force and moment coefficients basing on pressure coefficients obtained beforehand. By definition, the investigations of airfoil covers 2-D objects, thus only 2 force components (normal and tangential force) and 1 moment component (pitching moment) are needed to obtain.

Side force, roll moment and yaw moment are assumed zero. Lift coefficient is given by equation:

\[ C_L = C_N \cdot \cos \alpha - C_T \cdot \sin \alpha \]

where normal force coefficient \( C_N \) and tangential force coefficient \( C_T \) have been calculated by pressure coefficient distribution integrating:

\[ C_N = \oint_{\text{airfoil outline}} Cp \, d\tilde{x} \]
\[ C_T = -\oint_{\text{airfoil outline}} Cp \, d\tilde{y} \]

where \( \tilde{x} \) and \( \tilde{y} \) - airfoil coordinates, normalized by the chord length.

Pitching moment coefficient is given by equation:

\[ C_M = \oint_{\text{airfoil outline}} Cp \cdot (\tilde{x} - 0.25) \, d\tilde{x} + \oint_{\text{airfoil outline}} Cp \cdot \tilde{y} \, d\tilde{y} \]
3.2.6 CALCULATION OF REYNOLDS NUMBER

Reynolds number characterizes the conditions of measurement. This parameter is given by equation:

\[
Re = \frac{46.7 \cdot \rho_s \cdot Ma \cdot c \cdot \frac{T}{1 + 0.2 \cdot Ma^2} + 124}{\frac{1}{1 + 0.2 \cdot Ma^2}} \left( \frac{1}{T + 124 \cdot \frac{T}{273}} \right)^{1/2}
\]

where:
- \( p_s \) – static pressure [kPa]
- \( Ma \) – Mach number [-]
- \( c \) – chord length [m]
- \( T \) – temperature [K]

The above equation has been derived using Reynolds number definition:

\[
Re = \frac{\rho \cdot V \cdot c}{\mu}
\]

and Sutherland's formula – a dependency between viscosity and temperature of air:

\[
\mu = \mu_e \left( \frac{T}{T + 124} \right)^{\frac{3}{2}} \left( \frac{397}{T + 124 \cdot \frac{T}{273}} \right)
\]

4. RESULTS AND CONCLUSION

The discussed investigations encompassed 18 runs. An oscillation amplitude equals \( \Delta \alpha = \pm 5^\circ \) (in 16 runs) or \( \Delta \alpha = \pm 10^\circ \) (in 2 extant runs). Mach number equals 0.3, 0.4, 0.5 or 0.75.

An oscillation frequency varied from \( f = 0.3 \) Hz to \( f = 10 \) Hz, and base angle – from \( \alpha_B = -2^\circ \) to \( \alpha_B = 14^\circ \).

An analysis of investigation results has demonstrated that:
- A plot of lift coefficient \( C_L \) and angle of attack dependency have a loop shape. For lower values of angle of attack, where the flow separation on upper surface of the airfoil doesn’t exist the hysteresis loop is relatively narrow. A variation between steady and unsteady investigations’ results equals about \( \Delta C_L = 0.05 \) – both for increase and decrease of angle of attack.
- The width of hysteresis loop increases for greater base angles, as shown in Fig. 10.
In most cases, during angle of attack increase, greater values of lift and pitching moment coefficients than during its decrease (but for the same instantaneous value of α angle) have been achieved.

A plot of pitching moment coefficient $C_M$ and angle of attack dependency has a loop shape also, as shown in Fig. 11. For lower values of angle of attack the width of hysteresis loop is low as well.

An influence of Mach number and oscillation frequency on the width of hysteresis loop is very low.
• Maximal lift coefficient $C_{L_{\text{max}}}$ increases because of unsteady phenomena, and its increase exceeds $\Delta C_{L_{\text{max}}}=0.35$.

• Increase of oscillation frequency could cause an increase of critical angle of attack, up to $\Delta \alpha_{kr}=4.5^\circ$ for investigated airfoil.

• For high values of angle of attack, above critical angle of attack approximately, a steep decrease of pitching moment coefficient, up to $C_{M_{\text{min}}}=-0.2$, has been observed.

LITERATURE


Paweł Ruchała
Andrzej Krzysiak

METODYKA NIESTACJONARNYCH BADAŃ PROFILI ŚMIGŁOWCOWYCH

Streszczenie

Praca opisuje metodykę eksperymentalnych niestacjonarnych badań ciśnieniowych, dotyczących śmigłowych profili aerodynamicznych. Została ona wdrożona w trakcie badań profilu śmigłowcowego, wykonanych w Zakładzie Aerodynamiki Instytutu Lotnictwa dla PZL Świdnik i Ministerstwa Nauki i Szkolnictwa Wyższego (w ramach projektu celowego „Opracowanie i wdrożenie nowej generacji rozwiązań konstrukcyjnych, technologicznych i materiałowych dla wirnika nośnego i elementów płatowca śmigłowca PZŁ W-3A Sokół”).

Celem tego typu badań jest modelowanie w warunkach tunelowych zjawiska tzw. przeciągnięcia dynamicznego, które może wystąpić na łopatach wirnika nośnego śmigłowca w czasie lotu postępowego. Powoduje ono silne drgania łopat, wobec czego stanowi istotne ograniczenie osiągów śmigłowców. Zjawisko przeciągnięcia dynamicznego jest związane z szybką zmianą kąta natarcia, jaka występuje w czasie lotu postępowego śmigłowca. Warunki te odwzorowano za pomocą modelu wykonującego ruch oscylujący o zadanej częstości i amplitudzie. Mechanizm wywołujący oscylacje, jak również ujыта aparatura pomiarowa, zostały opisane w pracy.
Omawiana metodyka dotyczy badań ciśnieniowych. Opierają się one na wyznaczeniu rozkładów ciśnień na powierzchni przystosowanego do tego modelu.

Ze względu na nierównoczesność pomiarów, na podstawie ich wyników aproksymowano zmianę rozkładu ciśnień w funkcji czasu lub kąta natarcia. Zmianę współczynników siły nośnej i momentu pochylającego wyznaczono przez całkowanie tak wyznaczonego rozkładu ciśnień. Algorytm obliczeń również został omówiony.
Magnetoreological fluids as method for active controlling of landing gear shock absorber characteristic

Zbigniew Skorupka
Institute of Aviation

Summary

Smart materials are being used in much larger scale in mechanical solutions. Aviation usage of these materials seems to be natural because of interest in new technologies use in this industry. In this article authors discuss characteristics of magnetoreological fluids as a smart materials, examples of its industrial usage, requirements on landing gear characteristics, design and laboratory tests of model shock absorber in which MRF was used as damping fluid.

1. Magnetoreological fluid

Magnetoreological fluids (MRF) are qualified as smart materials, which are controlled by magnetic field. Every MRF consists of two fractions liquid and solid. Liquid fraction is mostly an oil (can be both synthetic or natural), which is responsible for MRF density when magnetic field is not applied. Viscosity of liquid fraction also has also limited interaction with speed of MRF response to magnetic field. Solid fraction of MRF is basically particles of ferromagnetic material mixed with liquid fraction. Solid fraction can be made in different sizes and different susceptibility to magnetic field. Generally MRF because of the composition (mixture of oil and ferromagnetic particles) has the tendency to sedimentation. Depending on the substance used as liquid fraction and the grains sizes of solid fraction sedimentation can be different in the same period of time and be less or more permanent. Some MRF can sediment in such manner that later mixing is impossible what makes their use in constructions (e.g. airplane shock absorbers) impossible.

The change of the property of the MRF consists in the change of the solid fraction particles arrangement in surrounding them liquid fraction. When no magnetic field is applied, the particles of solid fraction are arranged randomly in liquid fraction. When magnetic field is applied particles try to arrange according to lines of the magnetic field creating long chains (fig. 1, fig. 2). This results in the increase of density and shear strength.
Magnetic field, which is used to control the MRF, can be generated by any source from the permanent magnet to the coil. Controlling the behavior of MRF follows through the magnetic field value change. MRF reaction to changes in the magnetic field is immediate and allows to use MRF in applications where short times of the reaction are required.

MRFs are offered as the trade product by many companies. The largest and the most well known manufacturer of the MRF and MRF based devices is LORD Corporation.

**Tab. 1. Examples of Magnetoreological Fluids parameters by LORD Corporation [3]**

<table>
<thead>
<tr>
<th>Base Fluid</th>
<th>MRF-336AG</th>
<th>MRF-140CG</th>
<th>MRF-132DG</th>
<th>MRF-122EG</th>
</tr>
</thead>
<tbody>
<tr>
<td>Viscosity Pa·s</td>
<td>0.115 ± 0.015</td>
<td>0.280 ± 0.070</td>
<td>0.092 ± 0.015</td>
<td>0.042 ± 0.020</td>
</tr>
<tr>
<td>Density g/cm³</td>
<td>3.32-3.44</td>
<td>3.54-3.74</td>
<td>2.98-3.18</td>
<td>2.28-2.48</td>
</tr>
<tr>
<td>Solids Content by Weight %</td>
<td>% 82.02</td>
<td>% 85.44</td>
<td>% 80.98</td>
<td>% 72</td>
</tr>
<tr>
<td>Flash Point °C</td>
<td>&gt;150</td>
<td>&gt;150</td>
<td>&gt;150</td>
<td>&gt;150</td>
</tr>
<tr>
<td>Operating Temperature °C</td>
<td>-40 to +150</td>
<td>-40 to +130</td>
<td>-40 to +130</td>
<td>-40 to +130</td>
</tr>
</tbody>
</table>

2. USE OF MAGNETOREOLOGICAL FLUID

Unique properties of MRF create the possibilities of application in actively controlled energy dissipation devices. Example applications of MRF include:

- car suspension systems (fig. 3) ex. vibration dampers
- in pneumatic systems as the speed and position controller
- vibration damping and stiffness systems car seats
- devices reducing the results of strong wind and quakes (fig. 4)
- in artificial limbs for improved comfort
3. REQUIREMENTS FOR AIRPLANE LANDING GEAR AND SHOCK ABSORBERS

The main functions of the landing gear are:
- to absorb the kinetic energy of the vertical velocity
- to provide elastic suspension during taxiing and ground maneuvers
- to assure safety and comfort for passengers and transported goods during ground maneuvers, start and landing of the airplane.

These requirements are fulfilled in landing gear by properly designed shock absorbers and correctly chosen wheels.

There are some different landing gear designs that fulfill above criteria. In light airplanes dissipation of the energy can be achieved in different ways:
- spring L/G (in aeronautics L/G is an abbreviation for landing gear) made as a spring beam is both shock absorber and L/G strut (fig. 5)
- flexible elements the most often rubber or different elastomers built-up in the L/G structure
- steel ring springs
- oleo-pneumatic shock absorbers (fig. 6)
The first three solutions are preferred in light airplanes because of their low cost in connection with high efficiency rates and low weight. Yet use of the oleo-pneumatic shock absorber is the most effective solution.

Because of the largest efficiency in energy dissipation use of oleo-pneumatic shock absorbers is general in military and commercial airplanes where cost of the construction is not the most important criterion.

Oleo-pneumatic shock absorber absorbs energy by “pushing” a volume of hydraulic fluid against volume of gas (usually nitrogen but can be dry air.)

Oleo-pneumatic shock absorbers carry out two functions:
- a spring or stiffness function, which provides the elastic suspension by the compression of a gas volume
- a damping function, which dissipates energy by forcing hydraulic fluid through one or more small orifices

Fig. 7 Examplary shock absorber load versus stroke characteristic [1]

4. EXPECTATIONS OF MRF AS SHOCK ABSORBER FLUID EXPECTATIONS AND MRF CONTROLLING SYSTEMS

Use of MRF carries necessity of certain; quite serious changes in current shock absorbers or design new device initially capable of MRF use. In the case of the modernization of the older solution, there should be put special attention to different consistency of magnetoreological fluid. Also existence of solid fraction (iron particles) can be destructive for shock absorber parts (particularly seals). Another necessary change in shock absorber is to add one or more coils for MRF control. Design of entirely new shock absorber which is initially optimized for MRF use eliminates these problems, however this is not always possible.

Main goal of MRF use in airplane landing gear is possibility to correct shock absorber characteristics in order to achieve optimal energy dissipation in every landing condition. Naturally obtaining of the ideal profile is not possible for every landing because of general shock absorber design limits. Range of the possible L/G control is also limited by the safety reasons especially by requirement that landing must be safe even during MRF control power supply failure.

There are three methods of MRF based damping control.

First one is to put fixed damping characteristics into steering module (it can be both programmable computer or fixed hardware based logic). During landing phase, on board computer execute shock absorbers damping characteristic in order to achieve previously defined damping. Reference characteristic (put into MRF control system) is developed during laboratory tests of the shock absorber. That way of damping control enables two state steering:
- Without powered/executed MRF control system. In this case damping characteristic is only...
via mechanical systems built into shock absorber – passive damping

- With powered/executed MRF control system. In this case dumping characteristic is the one defined in electronic control system.

Main advantage of this control method is simplicity of MRF steering system, but there are only two damping characteristics available and energy absorption process is far from optimal. It is also ineffective because almost the same effect can be achieved by using standard well optimized hydraulic shock absorber.

Second way of MRF control (let’s call it semi-smart) is to create several damping characteristics optimized for different landing scenarios. Control system itself is more complicated than in first method because it has to act as databank for several characteristics and has to enable execution of right one. This can be achieved by onboard computer trigger based on existing instruments (without need to put external flight condition measuring systems). Semi-smart control system is better optimized than fixed one but still it is not the best solution.

Third and most advanced method of MRF control (let’s call it smart) is to enable real time control of shock absorber characteristic. This type of control enables full optimization of energy dissipation based on actual landing conditions. In order to achieve correct MRF steering it is need to provide fast computer system for damping characteristic calculation and set of high accuracy sensors which will measure parameters need to characteristic calculations. In most of the cases sensors for MRF control systems will be independent from airplane built-in sensors because of safety reasons (interferences, not enough accuracy, etc.). Execution of MRF steering is automatic and is performed by onboard computer; rest of the process is carried on by MRF control system. Smart system is most accurate of all three systems described but it is also most complicated and costly.

It can be assumed that for now semi-smart control system can be widely used in MRF based damping control due to reasonable cost/performance ratio.

5. MRF BASED SHOCK ABSORBER DESIGN

MRF based landing gear shock absorber was created by Warsaw Institute of Aviation Landing Gear Department within the project PBZ-KBN-115/T08/04 titled “Metallic, ceramic and polymer smart materials (design – obtainment – properties – use)”. MRF based shock absorber is a developed version of existing I-23 “Manager” small airplane front L/G. Necessary modifications were made to the existing design in order to use MRF but modifications didn’t change L/G layout.
First stage of the design change was to build MRF control coils. Coils were calculated using Finite Element Method (FEM) (fig. 9) by the FEMM [4] software.

Rys. 9 Example of FEMM programme based coil magnetic field analysis

After preliminary calculations of coils, which were crucial for shape of the rest of the design, it was 3D modeled. The design process was made by the use of the SolidEdge [5] software for 3D (fig.8) modeling and flat drawing. 3D modeling software was used in order to avoid structural mistakes and to speed up design process.

Shock absorber design process was aided by strength analysis using Femap (with Nastran calculating module) Finite Element Method (FEM) software.
That type of software significantly speed up design process and gives the chance of optimization before actual manufacture of the product. Finally prototype of MRF shock absorber was built (fig. 10). LORD MRF-336AG was used in design as a working fluid. Chosen MRF meets hydraulic oil flow requirements and can be easily mixed when sedimentation occurs.

Fig. 10 Prototype of MRF based shock absorber – internal parts

6. MRF BASED SHOCK ABSORBER LABORATORY TESTS

The tests of MRF shock absorber took place in Institute of Aviation Landing Gear Department Laboratory. Both static and dynamic test were made. Static tests were conducted in order to make shock absorber static characteristic. Dynamic tests were made in order to check shock absorber behavior in similar to actual conditions.

Static tests were conducted at 40T hydraulic press (fig. 11), dynamic tests were made on 10T drop machine (fig. 12).

Fig. 11. 40 T press for L/G static tests (picture:IA archive)  
Fig. 12. 10 T drop machine for L/G dynamic tests (picture IA: archive)

During static tests shock absorber was optimized in order to obtain the best possible profile for MRF flow control.
The dynamic tests of the shock absorber were executed on the object which static profile is showed on the graph below (profile Pr_810, fig. 13).

**Fig. 13. Static characteristic of MRF shock absorber (picture: IA archive)**

**Fig. 14. Vertical force versus shock absorber deflection for dynamic tests made with two control currents (both tests made with the same energy) (picture: IA archive)**

Dynamic test of L/G is a free drop of defined mass from fixed height in order to achieve desired landing energy.

Dynamic tests I23m257 (fig. 14) and I23m258 (fig. 14) were executed for the same reduced mass and L/G fall (the same energy), with relief of 2/3 dropped mass weight. During first dynamic test coils were not powered, current I=0[A]. In this case (I=0[A]) shock absorber design provide only hydraulic damping by pumping MRF through damping holes as in classic shock absorber solution. Shock absorber hydraulic damping was optimized in order to achieve the same damping value as in standard (non MRF based) I23 shock absorber.

During second dynamic test coils were powered by the current I =2[A]. Maximal L/G load decreased from 1376 [daN] in case of lack of current to 1218 [daN] in case of the 2[A] powered coils. There was about 11,5% decrease in load compared to the hydraulic damping itself (I=0[A]). Load change is a result of increased damping without change in other parameters of dynamic test.

7. SUMMARY AND CONCLUSIONS

Design of MRF based is no more complex than today’s standard solutions. There is no need of using additional control system because every modern airplane is equipped in on board power sources and computers that can be used as control electronics.

11,8% load reduction was achieved as the result of MRF use in shock absorber. Reduction was enough to conclude that shock absorber designs based on MRF could be made in future. Achieved effect was promising in spite of current was constant during drop tests. Active control of current can enlarge desired effect.
Use of proposed design can result in making, more reliable and better fatigue resistant landing gears. However high cost of MRF used during tests can put design to the economical test. But when one look at long term advantages of MRF based L/G such as improved safety and comfort designer should consider MRF based design as future of landing gear system.

LITERATURE


Streszczenie

Materiały inteligentne (ang. smart materials) znajdują coraz większe zastosowanie w konstrukcjach inżynierskich. Wykorzystanie ich w lotnictwie jako jednej z najbardziej nastawionej na nowoczesne rozwiązania gałęzi inżynierii jest jak najbardziej naturalne. Praca zawiera krótką charakterystykę cieczy magnetoreologicznej jako materiału inteligentnego, wymagania stawiane podwoziom lotniczym, opis konstrukcji oraz badań modelowego amortyzatora wykorzystującego MRF jako czynnik roboczy.
TEST RIG PROTOTYPE FOR TESTING OF NOSE LANDING GEAR ELECTRICAL ACTUATOR DESIGNED FOR PASSENGER AIRCRAFT

Zbigniew Skorupka, Wojciech Kowalski, Rafał Kajka
Institute of Aviation

Summary
Due to ecologic and economic reasons modern industry uses electrical drives and steering systems. Aviation is in this trend but restrictive safety and reliability requirements are the reasons of slower progress than in rest of the industry. In this article authors want to show process of designing, making and implementing test rig prototype for testing electrical actuator designed for front landing gear steering system of passenger aircraft.

1. AIRCRAFTS ELECTRICALLY ACTUATED SYSTEMS
In modern engineering solutions electric drives are replacing older hydraulic and pneumatic systems. It is natural process due to computer based control systems which are based on electric signals processing and analysis. Hydraulic and pneumatic control systems require special (dedicated) mechano-electric interfaces in order to enable computer based control. Some of the elements of hydraulic or pneumatic systems are simple to direct to control via computers but some of them (one can say that most of them) are controlled indirectly by interfaces mentioned above what can cause not satisfactory performance and accuracy of driven object most often compared with too high mass of the system.

In recent years ecological aspect is also taken into account. Oil based systems are extremely pollutive because of oils used as working fluids (problems with oil leakage, waste and utilization). In aviation SKYDROL oil is commonly used due to wide range of temperature in which it preserve nominal parameters but is very harmful for people managing it as well as for the environment.

Due to problems mentioned above, electrical systems are natural successor of current fluid and air based systems. In aviation electrical systems are tested for many years thanks to what many systems in airplanes are now electric based on servomechanisms used for steering flaps, ailerons, rudders or slots.

Big machines require heavy duty landing gears with efficient actuating systems in order to control them (retraction, brakes, nose wheel steering). Such parameters are now available by use of hydraulic systems.

Landing gear engineers are working on reliable and efficient electric steering system for passenger aircrafts nose landing gear wheels. Such work need to be done due to fact that in near future only landing gear will need hydraulic power system. Such configuration will be too heavy and too expensive for airplane owners. Also ecological aspect of electrical systems is a factor for landing gear control system change.
2. TEST RIG GENERAL REQUIREMENTS

Due to problems shown above, consortium was created in 2005 out of European aviation, SME and universities. Consortium successfully gained UE founding for design and tests of electrical control system for Airbus A-320 nose landing gear (fig. 1).

![Fig. 1. Airbus A-320 (source Airbus)](image)

Project was named „Distributed and Redundant Electromechanical nose gear Steering System“, acronym DRESS. The consortium was formed out of companies such as AIRBUS, SAAB, MESSIER BUGATTI, MESSIER DOWTY, EQUIPEARO, INSTITUTE OF AVIATION and others. Institute of Aviation role was to design and manufacture test rig for testing electric actuator for A-320 nose landing gear.

Range of tests to be made was wide. It consisted series of tests which purpose were to simulate as many scenarios as possible during landing gear operation. Tests were divided into two groups: dynamic and quasistatic. Range of parameters for each group is shown in table 1.

Dynamic tests consisted of rotation about L/G vertical axis with high speed rotation of L/G wheels (ex. just after touchdown) and added imbalance to the wheels which simulated tyre damage or runway roughness.

Quasistatic tests consisted of (rotation about L/G vertical axis):
- low frequency rotations to the desired position
- opposition of rotation from desired position

These tests were designed to simulate the resistance between the wheels and the airstrip surface, occurring during aircraft ground maneuvers. Additional condition was the possibility of implementation of the both groups of tests for various shock absorber deflections.
Due to above requirements it was necessary to design universal test rig. Universality requirement could not be met with a rigid structure due to the radically different motive needs for two groups of tests. It was decided to use a test rig modular design with common basis. As basis main frame with electric actuator; movable support and fixed bearing support was used. Interchangeable modules for dynamic (DCSS) and quasi-static tests (ATCSS) were mounted to the frame and to the shaft which is extension of dummy landing gear used in tests (fig. 2).

3. TEST RIG CONCEPT

The design of the test rig had to meet several requirements of the test procedure, safety requirements and have sufficient durability needed to carry out researches. First, a test rig must be designed in order to be neutral in the process of research its characteristics did not influence the obtained tests results. It was to be obtained for example by choosing such a structure in order to move test rig natural frequencies outside the measurement range of the test object parameters. At first it was thought that test rig should be designed in configuration that would reproduce real L/G mounting conditions i.e. to put L/G in vertical position as it is mounted in the airplane. This design was abandoned due to inability to reduce test rig natural frequencies to the desired level (not neutral for tests results) and due to not enough vertical space in target laboratory. Also operation and maintenance of such configuration could be too difficult.

After examination of the horizontal test rig configuration (fig. 2), it appeared that the change in orientation from vertical to horizontal simplifies design, increases the safety of operations, allows to meet the condition of natural frequencies and does not significantly affect the quality of tests results.
4. TEST RIG DESCRIPTION

As it was mentioned above, test rig was designed as modular construction consisting of three modules. First module consists of main (fixed) frame with two supports and dummy landing gear with electric actuator mounted to it. First support is movable in order to enable changes in shock absorber deflection. This support is also landing gear mounting base what resulted with necessity of adding stiffness representative which simulates real stiffness of L/G mounting in actual A-320.

Second support is used as base reference for whole test rig assembly. It is also mechanical safety module against excessive shaft rotation during tests (maximal shaft rotation is defined for each series tests separately). There are also sensors mounted to the test rig base module. These sensors are for acquiring parameters needed for test rig monitoring purposes as well as for actuator test results analysis.

4.1. QUASISTATIC TESTS MODULE – ATCSS

Quasistatic tests module (ATCSS) consists of hydraulic motor, backlashless clutch (which connects hydraulic actuator shaft with dummy L/G shaft) and hydraulic control system. After analysis of different actuators, it turned out that only the hydraulic motor is able to provide the required tests performance. Use of hydraulic motor was necessary because of huge amount of torque needed in tests as well as for stability of torque during tests with rotations in radial range of 180o (90o each side). Worth noting is the fact that the target user of the system wanted that the hydraulic installation is adapted to the hydraulic fluid “SKYDROL” commonly used in aviation because of the small parameters scattering due to the temperature changes.

It is necessary to remember that “SKYDROL” is an extremely aggressive liquid and requires
a special types of hydraulic (pipes made of special rubber and stainless steel, special types of seals) and mechanical parts (special coating paint which is resistant to corrosive factors.)

4.2. DYNAMIC TESTS MODULE – DCSS

Dynamic tests module (DCSS) is composed of an electrical acceleration system (with the asynchronous motor controlled by inverter and aviation grade wheel as drive for dummy L/G) mounted to the frame and driven discs mounted on a shaft of dummy L/G. Discs simulate wheels of a real aircraft. Discs can be equipped with unbalance masses on various radiuses. Imbalances accompanied with radial speed of the discs can produce various levels of torque excitations used for dynamic stability and durability tests of electric actuator (table 1.).
4.3. CONTROL SYSTEMS

Test rig control system consists of two electrical cabinets. In first of them, called Utility Bay (UB), there are all direct control devices for test rig and electric actuator are located.

**UB devices:**
- high and low voltage power supplies
- test rig sensors signals conditioners
- electromechanical devices for control and safety systems (ex. relays)

Utility Bay is also used for acquiring and send test rig signals to PXI computer control system, mounted in second cabinet called Control Command Bay (CCB). PXI computer is acting under the control of real-time operating system and is used for analyzing signals acquired from the test rig and for generating control signals based on acquired ones.

PXI computer is connected to the regular PC computer with installed LabView application for visual control of the test rig (MMI – Man Machine Interface, fig. 5). Block diagram of control system is shown in Figure 6.
5. TEST RIG MANUFACTURE

Manufacturing process of the test rig was not easy task because of its size and required manufacturing precision of the test rig components. Mechanical parts of the test rig were made entirely in Polish factories. The existing manufacturing base is entirely sufficient to perform even project like this one.
Fig. 8. Example graph of ATCSS test \((p=80\text{bar}, \text{torque}=2000\text{Nm}, \text{PID \{0,005 ,0 ,0\}})\)

Fig. 9. Example graph of DCSS test \((n_{\text{max}}=1150 \text{rpm}, \text{free run out, unbalance masses } m=2.5\text{kg}, r=0.27\text{m}, \text{automatic acceleration, mechanical blockade on mechanical limiter})\)
Assembly of the test rig was performed entirely by Landing Gear Department staff. Comprehensive staff was able to meet this challenge as well as it was able to cope with design of the test rig. Electrical and hydraulic control systems were manufactured in cooperation with specialized companies. However, the design of both control systems was made by Landing Gear Department staff.

Software, as well as control systems, was written by co-operating firm based on the detailed design and guidelines made by Landing Gear Department staff. The software has been optimized for the target user in accordance with his recommendations.

6. TEST RIG ACCEPTANCE TESTS

Assembled test rig was put under several acceptance tests. First stage of tests was a safety audit performed by Bureau Veritas in order to check compliance with CE markings required in target laboratory.

The next step of acceptance procedure was to check all functionality of the test rig before performing any tests. There were performed tests for DCSS module safety and functionality (e.g., spinning electric motor, checking if drive and driven wheel is connected well). In ATCSS module operation of all valves, the direction of movement of the hydraulic motor, the presence of leaks was checked.

After test rig check in manual mode, calibration tests were performed for both modules. During calibration tests dummy landing gear rotation was locked due to safety of the electric actuator which at the time wasn’t complete (control modules weren’t ready then). Electric actuator control modules were to be installed in target laboratory after test rig delivery.

Because test rig wasn’t mounted permanently to the ground, it was impossible to carry out all tests of DCSS module due to vibrations generated by the unbalanced masses mounted to the DCSS discs.

Acceptance tests were successful but it was necessary to make some improvements on the test rig. Main modifications were made in DCSS by changing electric motor acceleration curve, due to high power consumption during acceleration phase.

The tests also provided information on the functionality and parameters of the test rig various components what can help to carry out the numerical simulations of test rig behavior. Such simulation can be used during design process of another test rig.
7. CONCLUSIONS

Test rig for testing nose landing gear electric actuator was a great challenge for Landing Gear Department engineers. Test rig level of complexity due to size of tested object, demanding tests requirements and problems with finding manufacturers of such non-standard components made this project good opportunity to test design and logistic skills of Landing Gear Department staff.

Electric actuator test rig was crucial component of DRESS project due to necessity of reference tests results made on actual device to numerical analysis made by other consortium partners. Data comparison (from tests and simulations) will enable in the future of making more optimized electric actuator for another types of airplanes.

Thank to experience gained during project, IA Landing Gear Department is able to design and implement more test rigs not only for internal purposes but also for external clients. Fully assembled test rig is shown in the figure 5. In the same picture there is also IA Landing Gear Department team which made the test rig possible.

Fig. 10. Landing Gear Department team with fully assembled test rig (source IA)

REFERENCES

PROTOTYP STANOWISKA DO BADAŃ ELEKTRYCZNEGO AKTUATORA PODWOZIA PRZEDNIEGO DUŻEGO SAMOLOTU PASAŻERSKIEGO

Streszczenie

Ze względu na ekologię oraz ekonomię wszystkie dziedziny mechaniki coraz częściej korzystają z napędów i sterowania elektrycznego. Lotnictwo nie jest poza tym trendem, jednakże ze względu na konieczność zapewnienia niezawodności i bezpieczeństwa, prace nad wykorzystaniem układów elektrycznych postępują wolniej niż w innych gałęziach przemysłu. W poniższym artykule, autorzy pragną przedstawić proces projektowania i wdrożenia stanowiska do badań elektrycznego układu sterowania podwoziem przednim dla dużego samolotu pasażerskiego.
DETERMINATION OF NON-LINEAR AERODYNAMIC CHARACTERISTICS OF AN AIRCRAFT USING A POTENTIAL FLOW MODEL AND VISCOUS AIRFOIL CHARACTERISTICS

Janusz Sznajder
Institute of Aviation

Abstract
The article presents a hybrid method of determination aerodynamic characteristics of an aircraft at high angles of attack, consisting of a composition of a low-order panel method and modified Vortex Lattice Method. The modifications include determination of the position of control point for boundary condition based on two-dimensional lift slope of a wing section and an iterative procedure of simulating decrease of velocity circulation in a wing section due to flow separation through reduction of sectional angle of attack. The input data include two-dimensional aerodynamic viscous characteristics of wing sections along the wingspan. Since two-dimensional viscous airfoil characteristics can be computed with relatively low cost, or may be known from earlier wind-tunnel investigations, the presented method is very efficient at early design stages. The method is capable of analysing configurations with high-lift devices such as flaps or slats. The results of its application for the tailless configuration of PZL M-18 aircraft show good agreement of computed $c_{L_{\text{max}}}$ and $c_{L_{\text{max}}}$ for cruise configuration. For the landing configuration the $c_{L_{\text{max}}}$ coefficient is slightly underpredicted, while $c_{L_{\text{max}}}$ is predicted correctly.

1. INTRODUCTION
The region of aerodynamic characteristics at high angles of attack is very important for the successful design, since the value of $c_{L_{\text{max}}}$ determines stall speed of an aircraft, which is dictated by airworthiness requirements. This makes it important to be able to predict values of $c_{L_{\text{max}}}$ and $c_{L_{\text{max}}}$ reliably, on early phase of the design process, and for all configurations with different settings of high lift devices. Determination of aircraft aerodynamic characteristics at high angles of attack is a demanding task for numerical methods. Solutions of Reynolds-Averaged Navier Stokes solvers in this range of angles of attack depend strongly on the selected turbulence models and require a high-quality computational mesh which requires high workload and experience of the user of the software. A more economic option in terms of license costs and human workload is to use panel methods applying viscous-inviscid interaction, with boundary layer flow modeled by Prandtl equations.[1]. These methods allow for determination of friction drag, position of laminar-turbulent transition and separation of flow on airfoil and wing surface, but their accuracy in the region near $c_{L_{\text{max}}}$ decreases (Fig.1) due to quasi-two dimensional model of the boundary layer flow (neglecting spanwise flow and its influence on...
DETERMINATION OF NON-LINEAR AERODYNAMIC CHARACTERISTICS OF AN AIRCRAFT USING...

2. DESCRIPTION OF THE COMPUTATIONAL METHOD

An alternative method for determination of nonlinear wing characteristics at early design stage was proposed by van Dam [2]. In this method the spanwise distribution of circulation is determined using lifting line theory and modified using two-dimensional $\alpha$-$C_L$ characteristics of local cross-sections. The idea of modifying spanwise distribution of circulation presented in [2] is a basis for a present method merging panel method approach for modeling thick bodies[3] (fuselage, engine cowling, etc) with modified vortex lattice method adopting solutions presented in[2]. Combining of panel method approach with vortex lattice method makes it possible to determine pressure distribution on thick bodies and chordwise and spanwise load distribution on wings – an improvement over the original method[2], with the same capabilities at high angles of attack (Fig.2). The main assumption of the present method, as in[2] is, that by taking advantage of two-dimensional viscous characteristics of local airfoils it is possible to determine the spanwise distribution of circulation also for the range of angle of attack, for which there exist regions of separated flow on wing. The effects of high lift devices is taken into account in two-dimensional local characteristics which serve as a basis for modification of the spanwise distribution of circulation, determined for inviscid flow. An advantage of this approach is the possibility of using viscous characteristics obtained from wind-tunnel tests. The decrease of circulation, due to flow separation is modeled in the inviscid method by reduction of local angle of attack of an wing strip.

*Fig.1. Comparison of $-C_L$ curves of a single-engine aircraft computed with panel method applying viscous-inviscid interaction with results of wind tunnel experiment and flight test[1].*
2a. MATHEMATICAL MODEL

It is assumed, that flow is inviscid, irrotational, compressible and may be modeled with Prandtl-Glauert equation:

\[ (1 - M_c^2) \frac{\partial^2 \Phi}{\partial x^2} + \frac{\partial^2 \Phi}{\partial y^2} + \frac{\partial^2 \Phi}{\partial z^2} = 0 \]  

which may be reduced to Laplace equation

\[ \nabla^2 \Phi = 0 \]  

through a transformation of the reference system.

where \( \Phi \) is velocity potential.

The flow is modeled through a small disturbances of the free-stream velocity potential produced by surface distributions of singularities – sources and doublets. A constant-intensity doublet distribution is an equivalent of vortex ring placed on the boundaries of surface panel.

For thick bodies an internal Dirichlet boundary condition of constant, zero potential is applied [3]:

\[ \Phi = -\frac{1}{4\pi\sigma} \int_{\Sigma} \left( \frac{1}{r} \right) \left( \frac{\mu_i \cdot \nabla}{r} \right) dS = \frac{1}{4\pi\sigma} \int_{\Sigma} \mu_i \cdot \nabla \frac{1}{r} dS = 0 \]  

On wing surface the Neuman boundary condition of zero normal velocity is applied at control points:

\[ \vec{v} \cdot \vec{n} = 0 \]

The Kutta-Joukowski condition of finite velocities at trailing edge is fulfilled by the addition of vortex wake elements of circulation intensity equal to the wing vortex ring intensity at trailing edge.

The modification of the standard vortex lattice method is done in two steps. In the first step the slope of the linear part of the ‘inviscid’ \( c_L(\alpha) \) characteristic of local cross-section of the wing is corrected in order to be the same as the slope of the local airfoil viscous characteristic. This is done by the shift of the control point from standard position in 0.75 panel chord to a new position, which may be obtained by expressing the boundary condition (4) with induced velocity generated by the panel’s attached vortex of circulation intensity related to local \( c_L(\alpha) \) slope through Joukowsky theorem[2,3].

The second step concerns modeling of the effects of strong separation and extending the applicability of the method up to the negative slope of the \( c_L(\alpha) \) curve. The effect of flow separation is the reduction of velocity circulation at the local spanwise position. This is modeled with an iterative procedure of reduction of local angle of attack in wing strips where the computed value of \( c_L_{inviscid} \) exceeds the local airfoil’s \( c_L_{viscous} \) for the local angle of attack, determined as \( \alpha_{loc} = c_L_{inviscid} / a_{viscous} \), where \( a_{viscous} \) equals the local slope of \( c_L(\alpha) \) curve applied in the first step for the modification of the position of wing strip control points. In such situation the local angle of attack is reduced to a value, for which the new ‘inviscid’ \( c_L \) equals the ‘viscous’ \( c_L \) determined in the first step. The iterative procedure is necessary, since change of vortex circulation on one panel modifies values of induced angle of attack in all wing strips. In case when \( \alpha_{loc} \) determined in the first step exceeds \( c_{L_{max}} \) the new \( \alpha_{loc} \) is selected in the region of positive \( c_L(\alpha) \) slope. The new, lower value of local angle of attack leads to lower intensity of circulation in the next iteration, determined based on the boundary conditions. The algorithm of the iterative procedure of modification of circulation intensity in wing strips is shown in Fig. 2.
Fig. 2. Application of a hybrid panel/VLM method on aircraft surface
In order to test the method, it was applied for the determination of the aerodynamic characteristics of the PZL-M18 aircraft in tailless configuration including characteristics at high angles of attack. The model of PZL-M18 was particularly convenient for the test, because in wind tunnel test the two-dimensional airfoil characteristics and three-dimensional aircraft characteristics were determined for the same Reynolds number (this aircraft has a constant-chord wing).[4,5] In this case the method’s capabilities of accounting for different settings of high-lift devices could also be tested.

2b. RESULTS OF THE COMPUTATIONS

The airfoil characteristics with flap retracted and extended 30° in landing configuration were determined in wind-tunnel investigations conducted for \( M=0.2 \) and \( Re=1.4 \ mln \). The wind-tunnel investigations were element of the program of improving the aircraft’s high lift system effectiveness at low flight speed[6].

In the numerical test of the present method the three-dimensional flow around fuselage was not modeled. It was justified by the good agreement of the wing \( c_L \) at low angles of attack obtained with VLM method with the results of the wind-tunnel test for the wing-fuselage configuration. (In engineering methods of determination of aircraft characteristics it is often assumed, that \( c_L \) of central section of isolated wing is approximately equal to fuselage \( c_L \)). The central wing strip representing fuselage was excluded from the iterative procedure of modification of local angle of attack. The aircraft geometry and the geometry of the computational model is shown in Fig.3.

The evaluated values of \( c_L(\alpha) \), \( c_M(\alpha) \) i \( c_D(\alpha) \)compared with the results of wind-tunnel investigations are shown in Figs. 4,5,6.
Fig. 3. Geometry of the PZL-M-18 aircraft model in wind-tunnel investigations and model of wing surface applied in the present method
Fig. 4. Comparison of $c_{L(\alpha)}$ characteristics computed with the present method computed for two flap settings with results of wind-tunnel investigations [5]

3. CONCLUSIONS

A method for determination of aircraft non-linear, high-lift characteristics, based on modified Vortex-Lattice Method and Panel Method has been proposed and tested on a case of wind-tunnel model of a typical general-aviation low-wing aircraft using results of wind-tunnel experiment.

The results show a very good agreement of lift and moment characteristics at high angles of attack for the cruise configuration and slightly worse for a landing configuration with large flap deflection, but with proper prediction of $c_{L_{\text{max}}}$ due to flap deflection and for positive values of which is an operational range of angle attack.

The advantage of the present method is, that prediction of high-angle-of-attack aerodynamic characteristics of complex, three-dimensional configuration may be conducted using reliable two-dimensional characteristics which may come from experiment, or from easier to conduct, two-dimensional computations of viscous flow. Possible direction of development of this method is application for wings with moderate sweep, taking advantage of simple sweep theory.

LITERATURE


Streszczenie
Artykuł prezentuje hybrydową metodę wyznaczania charakterystyk aerodynamicznych samolotu na dużych kątach natarcia. Omawiana metoda jest złożeniem metody panelowej niskiego rzędu i zmodyfikowanej metody siatki wirowej. Modyfikacje polegają na wyznaczaniu położenia punktu kontrolnego warunku brzegowego metody siatki wirowej w zależności od nachylenia dwuwymiarowej zależności współczynnika siły nośnej w danym przekroju skrzydła od kąta natarcia oraz na zastosowaniu iteracyjnej procedury symulacji redukcji cyrkulacji w przekroju skrzydła będącej skutkiem oderwania opływu przez redukcję lokalnego kąta natarcia. Dane wejściowe zawierają dwuwymiarowe lepkie charakterystyki profili skrzydła wzdłuż rozpiętości. Ponieważ dwuwymiarowe charakterystyki mogą zostać wyznaczone numerycznie przy relatywnie niskim koszcie obliczeniowym lub też znane z badań tunelowych, przedstawiana metoda jest bardzo przydatna na wczesnym etapie projektowania. Metoda może służyć do analizy konfiguracji z urządzeniami zwiększającymi siłę nośną takimi jak klapy lub sloty. Wyniki uzyskane przy jej zastosowaniu do analizy konfiguracji bez usterzenia samolotu PZL M-18 wykazują dobrą zgodność wyznaczonych wartości $c_{L_{\text{max}}}$ i $\alpha c_{L_{\text{max}}}$ konfiguracji przelotowej. Dla konfiguracji do lądowania wyznaczony współczynnik $c_{L_{\text{max}}}$ jest lekko zawyżony, podczas gdy $c_{L_{\text{max}}}$ jest wyznaczone poprawnie.
APPLICATION OF A PANEL METHOD WITH VISCOUS-IN-VISCID INTERACTION FOR THE DETERMINATION OF AERODYNAMIC CHARACTERISTICS OF CESAR BASELINE AIRCRAFT

Wieńczysław Stalewski
Janusz Sznajder
Institute of Aviation

Definitions

$\alpha$, alpha angle of attack
$\beta$ angle of sideslip
$\delta$ aileron deflection
$\delta_a$ elevator deflection
$\delta_f$ flap deflection
$\delta_r$ rudder deflection
$\rho$ air density
$c_{L}, c_{M}, c_{N}$ moment coefficients about X, Y and Z aircraft axes
$c_{X}, c_{Y}, c_{Z}$ force coefficient in X, Y and Z aircraft axis direction
$c_{L}$ lift coefficient
$c_{D}$ drag coefficient
$P$ angular velocity component along aircraft X axis
$Q$ angular velocity component along aircraft Y axis
$R$ angular velocity component along aircraft Z axis
$U$ aircraft linear velocity component along aircraft X axis
$V$ aircraft linear velocity component along aircraft Y axis
$W$ aircraft linear velocity component along aircraft Z axis
$\bar{V}_{\infty}$ free stream velocity
$M_{\infty}$ free stream Mach number
$\phi$ disturbance velocity potential
$\Sigma$ surface of the body
$\Pi$ surface of the wake
$V_{t}$ transpiration velocity
$N$ unit normal to the surface
$\Theta$ momentum thickness
$\bar{H}$ transformed (compressible) shape parameter
$u_e$ tangent velocity at an external edge of boundary layer
$A, b$ matrix and right hand vector for the boundary layer equations system
Abstract

The article presents details of implementation of a panel method with viscous-inviscid interaction in an in-house developed code Coda3d. The code was applied for the determination of aerodynamic characteristics of baseline aircraft in a European Union 6-th Framework Program Cesar, aimed at acceleration of design and introduction to the market of light aircraft. The implemented physical model assumes that the flowfield around a flying object is divided into two zones. In the outer zone the flow is assumed inviscid, irrotational, compressible and may be modeled by Prandtl-Glauert equation. In the vicinity of the surface, up to the conventional border of the boundary layer the flow is modeled by a system of ordinary differential equations, which are derived from Prandtl boundary layer equations through integration in the direction normal to the surface. The system of boundary layer integral equations is then integrated in the direction of flow, separately on upper and lower surface of wing, starting from the stagnation point. The process of aligning of flow velocities on the outer border of boundary layer derived from viscous and inviscid models is conducted iteratively. The results of flow analysis include distribution of tangential velocities, pressure, friction drag coefficient and position of flow separation. Because of low computational cost and capabilities of estimation of viscous drag and aerodynamic characteristics at high angles of attack, the presented method is especially suitable for design and optimization process conducted in small enterprises.

1. INTRODUCTION

The goal of the CESAR project is a time-and-cost reduction of the whole process of design and market introduction of light general aviation aircraft in the category of 5-20 seats. Achieving of this goal is possible by introducing in early stages of design process efficient methods of multidisciplinary analysis and optimization concerning aerodynamic, mass and structural properties. The design process must take into account interactions between different design requirements and ensure modelling of physical phenomena affecting the performance, control and structural properties of the aircraft precisely enough for completion of the certification process, without the need for costly re-design of an aircraft in late stages of the design process.

The goals of the program set specific requirements for the computational tools applied in the design process. Since this project is addressed at the category of general aviation, the efficiency and low cost of computational tools are important for the effectiveness of the design process. In the field of aerodynamic design it is necessary to be able to predict the effects of high lift systems, estimate the drag forces of the configuration, including friction drag, and also to estimate the nonlinear characteristics at high angles of attack. Such requirements are fulfilled by Reynolds-Averaged Navier-Stokes equations solvers used in the design of large aircraft. In a case of small-aircraft development such approach is rather inefficient and prohibitively expensive because of the license costs and high workload associated with the CFD computational process. An alternative and attractive solution for this field of aeronautical design is application in the design process computational methods based on simplified flow model but giving relatively reliable results. The example of such approach may be panel methods coupled with boundary layer analysis using viscous-inviscid interaction technique. The cost of the whole computational process associated with panel methods (mesh generation, computations) is significantly lower than the cost associated with commercial Navier-Stokes solvers, with comparable accuracy of the solution within subsonic range of flights.

2. COMPUTATIONAL METHOD

The implemented physical model assumes that the flowfield around a flying object is divided into two zones. In the outer zone the flow is assumed inviscid, irrotational, compressible and may be modeled by Prandtl-Glauert equation:
\[
\left(1 - M_e^2\right) \frac{\partial^2 \phi}{\partial X^2} + \frac{\partial^2 \phi}{\partial Y^2} + \frac{\partial^2 \phi}{\partial Z^2} = 0
\]  

which may be reduced to Laplace equation
\[
V^2 \phi = 0
\]  

through the Prandtl-Glauert transformation of the reference system.

The above equation is solved using Boundary Element Method with Dirichlet, internal boundary condition. The following integral equation should be fulfilled by potential for every point lying on an internal side of body’s surface:
\[
\iint_{\Sigma} \phi \frac{\partial}{\partial N} \left(\frac{1}{r}\right) d\Sigma + \iint_{\Pi} (V_e \cdot N + V_t) \frac{1}{r} d\Sigma - \iint_{\Pi} (\Delta V_e) \frac{1}{r} d\Pi - \iint_{\Pi} \Delta \phi \frac{\partial}{\partial N} \left(\frac{1}{r}\right) d\Pi
\]  

where: \(\Sigma\) - body’s surface, \(\Pi\) - wake’s surface, \(V_t\) - transpiration velocity, \(N\) - unit normal to the surface.

Additionally, for lifting surfaces (e.g. wing, flaps, rudders), the potential should fulfill the Kutta Condition, expressing (in the presented approach) equality of pressures on the upper and the lower surface of trailing edge.

The flow in the vicinity of the surface, up to the conventional border of the boundary layer is modeled by Prandtl equations:

\[
\frac{\partial (\rho u)}{\partial s} + \frac{\partial (\rho v)}{\partial n} = 0
\]

\[
\rho u \frac{\partial u}{\partial s} + \rho v \frac{\partial u}{\partial n} - \frac{\partial \tau_{zn}}{\partial n} = \rho \frac{\partial u}{\partial t} + \rho v \frac{\partial u}{\partial s}
\]

These equations, integrated across boundary layer give the following system of ordinary differential equations:

\[
\frac{d\theta}{ds} + \left[2 + H - M_e^2\right] \frac{\theta}{u_e} \frac{du_e}{ds} = \frac{C_f}{2}
\]

\[
H \frac{d\theta}{ds} + \theta \frac{dH}{dH} \frac{d\bar{H}}{ds} + \left[1 - M_e^2\right] \frac{H \theta}{u_e} \frac{du_e}{ds} = C_E
\]

\[
H \frac{d\theta}{ds} + \alpha \frac{d\bar{H}}{ds} + \left[\alpha \left(1 + \bar{H}\right) + H \left(1 - M_e^2\right)\right] \frac{\theta}{u_e} \frac{du_e}{ds} = S^*
\]

In the above system the unknowns are integral parameters of boundary layer being one-argument functions of arch length \(s\) measured along wing or wake surface.

Additionally, the entrainment parameter \(C_E\) is modeled applying Green approach [1]. Based on theoretic-experimental relations between different parameters of the boundary layer, an additional differential equation for the \(C_E\) coefficient is derived:
This approach concerns only the lifting surfaces like wings, flaps, control surfaces having attached wake surfaces. For these surfaces, two-dimensional boundary layer equations are solved in chordwise sections of both the solid surfaces and the wakes. The applied integral methodology leads to the system of three ordinary differential equations on integral parameters of boundary layer. This system may be written in the following form:

\[
\frac{dC_E}{ds} = F(\theta, \overline{H}, u_e, \ldots)
\]

where:

- \(\theta\) is the momentum thickness, the transformed (compressible) shape parameter, \(u_e\) is the tangent velocity at an external edge of boundary layer.

The above general system of equations is solved using three different methods for three types of flow inside the boundary layer: laminar, turbulent-attached, turbulent-separated. The equations are integrated along “chordwise path lines”, starting from stagnation point, separately for upper and lower surface of the wing. During the calculations algebraic transition criteria are checked. After reaching the trailing edge, the calculations are continued along the wake. One of the results of the calculations is distribution of transpiration velocity, modelling the thickness of the boundary layer.

This transpiration velocity is used to modify the boundary conditions for panel method’s solution of the external flow problem in the outer sub-domain. The boundary layer calculations make it possible to assess the friction drag. It may be done either by the integration of friction coefficient along the body’s surface or by using the Squire-Young Formula based on integral parameters of boundary layer at far-wake position. For surfaces for which the boundary layer calculations are not performed (e.g. fuselage) the friction drag is evaluated using given, constant value of local friction coefficient, typical for given flight conditions.

Solution of the system of equations (10) may be conducted in two modes: direct or inverse. Due to numerical conditions [2], the direct mode is applied for the analysis of turbulent boundary layer in attached flow, while inverse mode is applied for regions with separated flow. In the direct mode it is assumed, that the chordwise distribution of tangent velocity \(u_e\) is known as a solution of the external inviscid flow problem.

In this case a three-equation version of system (10) is solved with equations (6),(7),(9) and unknowns \(\theta, \overline{H}, C_E\). For steady flows the system of equations (6),(7),(9) becomes singular, when \(dH_i/dH = 0\) [2]. This happens for \(\overline{H} = 2.7\).

Experimental data shows that for this value of parameter a separation of turbulent boundary layer takes place. Due to this singularity, computation in direct mode can not be continued. Further computations are conducted in inverse mode, solving the system (10) for four unknowns: \(\theta, \overline{H}, C_E, u_e\).

In this case it is assumed, that chordwise distribution of entrainment parameter \(S^*\) is known. According to [2] such system of differential equations for steady flow is not singular in the
separation region.

The result of the solution of boundary layer equations is the distribution of ‘viscous’ tangential velocity at the outer limit of the boundary layer. In general case the distribution differs from the distribution of inviscid’ tangential velocity determined as the solution of external inviscid flow. The aligning of ‘external’ and ‘internal’ tangential velocity is performed iteratively. In subsequent iterations the distribution of $S^*$ parameter is modified in order to minimize the difference of the distributions of tangential velocities. For aligning the distributions of ‘external’ and ‘internal’ tangential velocities it is necessary to evaluate the dependence of the $S^*$ parameter on the gradient of tangential velocity $u_e$. This dependence has the following form:

$$\frac{\partial u_e}{\partial S} = G_2 \cdot S^* + G_1 \quad (11)$$

where $G_1,G_2$ are functions of boundary layers parameters.

The viscous-inviscid interaction is modelled with a semi-inverse method, based on Le Balleur concept[3]. In regions with attached flow the interaction is modelled in a direct approach with the distribution of tangential velocity from inviscid flow as a boundary condition for the computations of flow in boundary layer. In these computations a distribution of transpiration velocity is determined, which modifies the boundary condition of the external inviscid flow.

The cycle of computations of inviscid and viscous flow is repeated until a converged solution is reached. For regions of separated flow a ‘semi-inverse’ approach is applied.

Regardless of whether the flow is separated or attached, the computations are performed in iteration cycles. In each iteration cycle the following steps are performed:

1. computations of inviscid external flow
2. computations of viscous boundary layer flow

Let’s assume that $(n+1)$-th iteration cycle is being performed. The following notation is applied:

- $u_e(n)$ - distribution of tangential velocity in $n$-th iteration cycle
- $V_e(n)$ - distribution of transpiration velocity in $n$-th iteration cycle
- $S^{*}_{(n)} = v_{(n)} / u_{e(n)}$

In the first step of each iteration cycle the inviscid external flow is computed with panel method for given boundary conditions. The most important boundary condition from the point of view of viscous-inviscid interaction is the distribution of transpiration velocity on a wing strip surface and on the vortex wake. In the first iteration zero-transpiration velocity distribution on wing surface is assumed. In the viscous-inviscid interaction problem the evaluation of external inviscid flow amounts to the evaluation of the distribution of the tangential velocity $u_{e(n+1)}$ on wing strip and vortex wake for given distribution of transpiration velocity $V_{e(n)}$.

In the second step of $(n+1)$-th iteration cycle the computations of boundary layer flow are conducted. The input data consists of:
- distribution of tangential velocity $u_{e(n+1)}$ computed in inviscid distribution of
- parameter for given boundary condition – transpiration velocity $v_{e(n)}$ in inviscid flow.

In the beginning of the computations of boundary layer flow the position of stagnation point (stagnation line on a wing) is determined based on the distribution of tangential velocity. Then the flow is divided between upper and lower surface. For each surface the computation of boundary layer flow is conducted separately.
Starting from the stagnation point, along the wing strip and further along the vortex wake, boundary layer equations are integrated step-by-step down the flow path. The integration starts from integrating of the laminar boundary equations in the region near the stagnation line. Continuously with the integration the criteria of laminar-turbulent transition are checked. In case of detecting the transition, or reaching a point of enforced laminar-turbulent transition, process of integrating turbulent boundary-layer commences. Initially, up to the point of possible boundary layer separation the computations are conducted in the direct mode. For given distribution of $u_{e(n+1)}$ tangential velocity, values of unknown boundary layer parameters: $\theta$, $\overline{H}$, $C_E$ are evaluated, and then, using eq. 12, new distribution of evaluated.

The final distribution of $S^*$ parameter is evaluated (for laminar and turbulent boundary layer) from the equation:

$$S^*_{(n+1)} = S^*_n + \sigma d \cdot \left( S^*_{v(n+1)} - S^*_n \right)$$

with $\omega_d$ being under-relaxation coefficient for direct mode.

In parallel to the computations of turbulent boundary layer the criterion of flow separation is checked. If the separation occurs, the computations downstream of the separation point are conducted in inverse mode. For given the unknowns evaluated.

Let’s assume, that $u_{eV(n+1)}$ is the distribution of tangential velocity determined in ($n+1$) iteration as the solution of boundary layer equations in inverse mode. Then the new distribution of $S_{(n+1)}^*$ in separated flow region is determined according to[2],[3]

$$S^*_{(n+1)} = \begin{cases} S^*_n + \omega_1 \cdot S^*_{e\text{d}(n+1)} & \text{dla } M_e < 1 \\ S^*_n + \omega_1 \cdot S^*_{e\text{b}(n+1)} & \text{dla } M_e > 1 \end{cases}$$

(14)

where:

$$S^*_{e\text{d}(n+1)} = \frac{\beta}{\Delta s} u_{e(n+1)} - \beta \cdot G_2 \left\{ \frac{1}{u_{eV(n+1)}} \frac{du_{eV(n+1)}}{ds} - \frac{1}{u_{e(n+1)}} \frac{du_{e(n+1)}}{ds} \right\}$$

(15)

$$S^*_{e\text{b}(n+1)} = \frac{\beta}{\Delta s} \left\{ \frac{\pi}{u_{e(n+1)}} \left( \frac{u_{e(n+1)}}{\Delta s} \right)^2 + \left( \beta \cdot G_2 \right)^2 \right\} \left\{ u_{e(n+1)} \left[ \frac{d^2 u_{eV(n+1)}}{ds^2} - \frac{d^2 u_{e(n+1)}}{ds^2} \right] \right\}$$

(16)

In the above equations $\omega_i$ is an under-relaxation coefficient for the inverse modes $\Delta s$ is a characteristic dimension of computational mesh, $G_2$ is derivative of tangential velocity gradient $d_{u_{e}}/ds$ with respect to $S^*$ parameter defined in Eq(11).

Finally, after the solution of boundary layer equations on upper and lower surface, new distribution of entrainment parameter $S^*_{(n+1)}$ is computed. Based on this distribution, new distribution of transpiration velocity $v_{e(n+1)}$ is evaluated. This distribution is a new boundary condition for the method computing external inviscid flow in new iteration cycle.

The computational cycle described above is conducted until convergent solution is found.
As a measure of convergence two norms were adopted:
- difference between distributions of velocity potential in two subsequent iterations
- difference between distributions of transpiration velocity in two subsequent iterations

In case when the two norms reach sufficiently low value, the iteration process is stopped.
The methodology described above was realised practically as a computational software named CODA3DPanel3dbl. The software is component of IoA in-house software package CODA3D which is a tool supporting aircraft design process.

2. COMPUTATIONAL COST OF PANEL METHODS

Computational cost associated with panel methods may be divided between the generation of the computational mesh and the process of computations. Such approach makes it possible to compare the computational cost of panel methods with costs of more advanced CFD solvers.

2.1. Generation of the computational mesh

The computations of flow are performed on the meshes created on the wetted surfaces and vortex wakes. Considering that the typical contemporary design process is conducted using CAD tools, it is necessary to apply in the mesh generation a dedicated software capable of reading CAD exchange format. This allows also the re-edition of the wetted surface in order to prepare it for meshing. The re-edition involves joining small surface elements into bigger segments, easier for meshing and to remove surface discontinuities, joints and small details (lamps, covers, antennas, small air-inlets). An additional task on this stage of the process is the definition of the vortex wake behind lifting and control surfaces, and, in the case of large air-inlets, the definition of inlet surfaces for setting the inlet boundary condition.

This task, and also the preparation of the computational mesh was done using a dedicated software - Gambit grid generator, developed by Ansys company. The preparation of the computational mesh was done using typical functions for the preparation of two-dimensional, structural grids. The Coda software - IoA implementation of the panel method - has also an additional software tool for fine-tuning the mesh created by other software. It is possible, for example, to increase mesh density in regions where the computations reveal strong pressure gradients.

Comparing the manual workload involved in generating mesh for panel methods and CFD RANS solvers, it may be estimated, that the time of preparation of the surface mesh for panel method is an order of magnitude lower than the time necessary to generate a three-dimensional mesh with high-quality boundary layer region.

2.2. Computations

Computational time and memory requirements are squarely proportional to the size of the problem. Typical computational mesh for single-engine aircraft in configuration with flaps extended is composed of roughly 20000 elements. The number of surface elements of the computational mesh determines the size of the matrix of influence coefficients (velocity potential induced in each panel's control point by each panel) which is number of panels squared. For a low-order panel method without viscous-inviscid interaction the time of computations depends mainly on time necessary for the computation of influence coefficients and time of solution of large system of linear equations. Enhancing the potential flow solution with viscous-inviscid coupling increases computational time. This time increase depends on the type of the viscous-inviscid coupling. For low-to-moderate angles of attack without large areas of flow separation there exists a week interaction between the boundary layer and the inviscid flow. This type of interaction may be analysed with a direct methods of boundary layer analysis (determination of transpiration velocity modifying the boundary condition of inviscid flow, based on distribution of tangential velocities from potential flow). Direct analysis of boundary layer is
a fast converging process and increases the computational time by 10-20% in comparison to calculations of invicid flow. For higher values of angle of attack, in the region of \( c_{L, \text{max}} \) the analysis of boundary layer must be conducted in inverse mode (evaluation of boundary layer characteristics and tangential velocity on the outer boundary based on transpiration velocity from the previous time step). This type of analysis is much more slowly converging, and the computational time may be increased by 50-100% of the computation time of inviscid solution.

This, however, is the price for obtaining an estimate of \( c_{L, \text{max}} \) and \( \alpha c_{L, \text{max}} \), which cannot be estimated in pure potential flow model. The increase of computational time depends also on the level of complication of the aircraft configuration. A cruise configuration is analysed faster than a landing configuration with large areas of separated flow on the surface of flaps. The analysis of such multi-element configuration may even require as much computational time as obtaining a Navier-Stokes solution with a commercial code.

**Tab 1. Comparison of computational effort in Panel Methods (PM) and CFD solvers**

<table>
<thead>
<tr>
<th>Application</th>
<th>PM without viscous-inviscid interaction</th>
<th>PM with viscous-inviscid coupling in direct mode</th>
<th>PM with viscous-inviscid coupling in inverse mode</th>
<th>CFD – Reynolds Averaged Navier Stokes equations</th>
</tr>
</thead>
<tbody>
<tr>
<td>linear range of ( c_L ), computation of lift and moments, induced drag</td>
<td>linear range of ( c_L ), computation of lift, moments, induced and friction drag</td>
<td>linear and non-linear range of ( c_L ), computation of lift, moments, induced and friction drag</td>
<td>linear and non-linear range of ( c_L ), computation of lift, moments, induced and friction drag</td>
<td></td>
</tr>
<tr>
<td>machine memory</td>
<td>~0.1</td>
<td>~0.1</td>
<td>~0.1</td>
<td>1.0</td>
</tr>
<tr>
<td>computational time</td>
<td>~0.1</td>
<td>~0.1-0.15</td>
<td>~0.2-0.3 up to 1.0</td>
<td>1.0</td>
</tr>
</tbody>
</table>

The advantage is, however, much lower size of the required machine memory (an order of magnitude) and faster and easier preparation of the mesh.

The increase of computational time depends also on the level of complication of the aircraft configuration. A cruise configuration is analysed faster than a landing configuration with large areas of separated flow on the surface of flaps.

The analysis of such multi-element configuration may even require as much computational time as obtaining a Navier-Stokes solution with a commercial code. The advantage is, however, much lower size of the required machine memory (an order of magnitude) and faster and easier preparation of the mesh.

### 3. APPLICATION OF IOA PANEL METHODS IN THE CESAR PROJECT

In the Cesar project the method described above was used in two tasks: Task 1.2 – Advanced Wing Design and Task 1.3 – Flight Dynamics. In this article the application of panel methods in Task 1.3 is described. The method was applied for the aerodynamic analysis of two baseline light transport aircraft. Both aircraft, single- and two-engine, designated AC1 and AC2 had capacity of about 10 seats. In case of single-engine AC1 aircraft experimental aerodynamic characteristics, obtained in wind-tunnel tests and in-flight tests were available for comparison. In case of the AC2 aircraft the experimental data was not available due to the project being on earlier stage of the design. The aerodynamic characteristics determined using the panel method were used as input data for further activities in Task 1.3. They were used in the development of numerical tools for the optimization of aircraft dynamic characteristics, and also as input data of another...
3.1. Computational meshes

The computational meshes were created based on geometric data encoded in IGES format – an exchange-geometric-data format in CAD systems. Due to aircraft vertical symmetry the surface meshes were created for one symmetric half of the configuration. A mesh comprising one symmetric half of the configuration is sufficient for the determination of aerodynamic forces acting in the XZ plane and the pitching moment. For the determination of forces acting in the spanwise direction and rolling and yawing moments a mesh comprising the whole aircraft is necessary. Such meshes were created through mirror reflection of one symmetric half of the configuration. The computational meshes were created partially using the Gambit grid generator – an element of the Fluent CFD solver.

The Gambit was used especially to create some meshes within difficult regions of aircraft surfaces like connections wing-fuselage, wing-engine-nacelle. Many meshes were created and configured using CODA in-house IoA software package. It concerns especially creation of artificial surfaces of wakes, not defined in IGES files describing aircraft geometry. For multi-element-wing configurations, vortex wakes were generated and attached to each component of wing high-lift system.

The applied IoA panel method enables to model deformable wakes. However, in presented calculations the wakes were not flexible, which means that they were created before the start of computations and were not modified during computations. For every wing or other lifting surface, the vortex wake surface was tangent to the wing mean surface near the trailing edge, and farther from the lifting element it is tangent to a plane formed by fuselage (X) axis and spanwise (Y) axis (Fig.1a, 1b).

Fig. 1a Wing vortex wake for the take-off configuration
3.2. Results of computations

The computations were performed for the cruise, take-off and landing configurations. The take-off and landing configurations differed from each other in positions and deflections of flaps. For the cruise configuration (flaps retracted) wings were modelled as one-element surfaces, without division into the main element and flap. This is justified by the stagnation of flow between the two elements, and this significantly speeds up the computation due to lower number of surface elements. The extended landing gear was ignored in the geometric model, due to inability of the potential flow model to resolve flow around non aerodynamic shapes. The propeller wake was also ignored.

1) Cruise configuration

Surface distribution of pressure coefficient on cruise configuration at alpha=0° is shown in Fig. 2. The cruise alpha-c\text{L}, alpha-c\text{M} and c\text{D} - c\text{L} characteristics are shown in Figs. 3 - 5 in comparison with wind-tunnel data and flight-measurements done during the Cesar project. The c\text{M} values were computed in reference to the point located in 25% of mean geometric chord.

The c\text{L} and c\text{M} values were obtained through the direct integration of pressure on aircraft surface, while the c\text{D} is a sum of the induced drag and local friction forces. Induced drag was calculated in the Trefftz plane and friction drag was determined for lifting surfaces (surfaces with attached vortex wake). For non-lifting surfaces (fuselage and nacelles) the mean value of friction coefficient determined for lifting surfaces was assumed. This procedure is the result of the constrains of the method of boundary layer analysis, which is limited to surface segments with attached vortex wake (wings, tail surfaces, etc.).
Fig. 3 Lift coefficient vs. angle of attack of the cruise configuration

Fig. 4 Moment coefficient vs. angle of attack of the cruise configuration

Fig. 5 Drag polar of the cruise configuration
The most important differences between the characteristics computed with the panel method and the experimental results are: over-prediction of \( c_{L_{\text{max}}} \) and \( \alpha c_{L_{\text{max}}} \) higher negative value of \( \frac{\partial c_m}{\partial \alpha} \) and lower values of drag.

There are several possible reasons for the over-prediction of \( c_L \) at high angles of attack: quasi-two dimensional model of the boundary layer flow (neglecting spanwise flow and its influence on the flow separation), effects of surface roughness and surface imperfections (rivets, surface segmentation etc.) and additional turbulence caused by the propeller wake. It should be pointed out, that accurate prediction of \( c_{L_{\text{max}}} \) and \( \alpha c_{L_{\text{max}}} \) is still a challenge for advanced RANS solvers, and the accuracy of results in the high-alpha range is usually lower than at lower values of angle of attack. Compared to the characteristics obtained from wind tunnel and flight experiment, the \( c_L \) characteristic computed with panel method has higher value of \( c_{L_{\text{max}}} \) and higher \( \alpha c_{L_{\text{max}}} \).

The small differences between \( c_L \) values at lower alphas are most likely due to different position of C.G. which causes different deflection of elevator needed to obtain \( c_M=0 \). This may also explain different position of minimum drag points on \( c_L - c_D \) polars computed for \( c_M=0 \) and obtained from flight experiment.

The different slope of \( \frac{\partial c_m}{\partial \alpha} \) is most likely the effect of different reference point. In Table 5 the values of \( c_{M\alpha} \) derivative computed with panel method are compared with results of a handbook method [5] and Vortex Lattice Method [5] with the reference point in 25% of mean geometric chord. The agreement between the results is better than in Fig.4 and the \( \frac{\partial c_m}{\partial \alpha} \) value computed with the panel method is less negative than the results of other methods.

The underestimating of drag values may be caused by computing the drag forces only on wing and tail surfaces, and scaling those values by the factor \( \left( \frac{S_{\text{wet}}}{S_{\text{wet\,wing+\,tail}}} \right) \).

This is caused by the limitations of the applied method of boundary layer analysis, which may be used only for surfaces with trailing edges and vortex wakes and it cannot predict effects of surface roughness and imperfections.

2) TAKE-OFF AND LANDING CONFIGURATIONS

The take-off and landing configurations were analysed using a symmetric half-model with 9727 surface panels. The flap settings were obtained directly from CATIA file provided by the Aero-Vodochody company. Boundary layer analysis was performed on either main element and flap segments. The vortex wake was attached to the trailing edge of each element (main-central, flap and outer-wing) and propagated downstream in a direction locally tangent to the bisection of airfoil trailing-edge angle. Further downstream the vortex lines were tangent to a \( XY \) plane. The distribution of pressure coefficient in take-off configuration is shown in Fig. 6. The \( \alpha-c_L \), \( \alpha-c_M \), \( \alpha-c_D \) characteristics are shown in Figs. 7-12. The \( \alpha-c_L \) characteristic has been shown in two variants: one for zero elevator deflection, and the second one for the elevator deflection angle set for balance condition in steady-level flight. The \( \alpha-c_M \) and \( c_D-c_L \) characteristics are determined for zero elevator deflection. The differences between the computed and experimental characteristics are similar as for the cruise configuration.
Fig. 6. Distribution of pressure coefficient on the surface of AC2 aircraft in take-off configuration at $\alpha=22^\circ$ (flow computations conducted for left symmetric half-model)

Fig. 7. Comparison of lift coefficient vs. angle of attack curves for the take-off configuration computed with panel method and obtained from wind tunnel experiments and from flight test
Fig. 8. Comparison of moment coefficient vs. angle of attack curves for the take-off configuration computed with panel method and obtained from wind tunnel experiments.

Fig. 9. Comparison of drag polars for the take-off configuration computed with panel method and obtained from wind tunnel experiments and flight test.
Fig. 10. Comparison of lift coefficient vs. angle of attack curves for the landing configuration computed with panel method and obtained from wind tunnel experiments and from flight test.

Fig. 11. Comparison of moment coefficient vs. angle of attack curves for the landing configuration computed with panel method and obtained from wind tunnel experiments and from flight test.
One of the results of the analysis of boundary layer is a position of the separation point along the span. This is determined based on change of sign of friction coefficient, from positive to negative (which indicates appearance of inverse flow). Computations of the flow in the flap region in landing configuration indicate large region of separation of flow on flap upper surface (40-50% of flap chord). This result is an indication that it may be possible to find flap settings enabling higher effectiveness of the flap for the landing configuration (Fig. 13).
4. DETERMINATION OF AERODYNAMIC DERIVATIVES

4.1. Force and moment derivatives with respect to linear and angular velocities

**Basic assumptions:**

1) The calculation of aerodynamic derivatives is based on aerodynamic characteristics obtained with panel method for steady, viscous, compressible flow. This is makes it possible to determine the values of derivatives with respect to small changes of linear and angular velocities. The aerodynamic derivatives with respect to aircraft accelerations can not be determined with the applied method. Their determination would require the analysis of accelerated flight and unsteady flow, which is out of scope of this work and would require much more computational resources. The only realistic methods for the determination of derivatives with respect to aircraft accelerations are so far wind-tunnel experiments and engineering data.

2) The aerodynamic derivatives are computed as differential quotients of the increments of forces (moments) and linear or angular velocities.

3) All aerodynamic forces, coefficients and velocities are defined in stability axes: X – in the vertical symmetry plane, parallel to the flow direction, directed forwards, Z – in the vertical symmetry plane, perpendicular to X, directed downwards, Y-perpendicular to X and Z, directed right.

**Definitions:**

\( \{V_a\} \) - velocity of the origin of aircraft coordinate system, \( \{V\} = \{U, V, W\} \)

- vector of aerodynamic forces and moments \( \{F\} = \{X, Y, Z, L, M, N\} \)
- vector of aircraft velocity disturbances (dimensional) \( \{D\} \)

\( \{D\} = \{u, v, w, p, q, r\} \)

- vector of characteristic dimensions:

\( \{l_i\} = \{1, 1, 1, b, c, b\} \) (17)

- vector of aerodynamic coefficients in stability axes:

\( \{C\} = \{C_x, C_y, C_z, C_L, C_M, C_N\} \)

- vector of derivatives of aerodynamic coefficients with respect to non-dimensional velocities [4]:
A dimensional aerodynamic derivative is defined as:

$$ \{C_y\} = \left\{ \frac{\partial C_i}{\partial D_j} \right\} $$  \hspace{1cm} (18)

A non-dimensional aerodynamic derivative is defined as:

$$ F_y = \frac{\partial F}{\partial D_j} = \frac{\partial}{\partial D_j} (0.5V_A^2 S_l C_i) $$ \hspace{1cm} (19)

A non-dimensional aerodynamic derivative is defined as:

$$ P_y = \frac{\partial}{\partial D_j} (0.5V_A^2 S_l C_i) \frac{1}{0.5ASV_A l_j l_j} $$ \hspace{1cm} (20)

in the stability axis system

$$ f_1 = 1, \quad f_2 = f_3 = f_4 = \ldots = 0 $$

in the body axis system

$$ f_1 = \cos \beta \cos \alpha, \quad f_2 = \cos \beta, \quad f_3 = \cos \beta \sin \alpha $$

A dimensional aerodynamic derivative may be obtained from the non-dimensional derivative through the equation:

$$ F_y = p_y \frac{0.5 AS l_i V_A}{(22)} $$

with:

$$ l_i $$ characteristic dimension, as in (17)

### Tab. 2. Definition of non-dimensional derivatives:

<table>
<thead>
<tr>
<th>i/j</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$2 C_{x1} + C_{xu}$</td>
<td>$2 C_{x2} + C_{xv}$</td>
<td>$2 C_{x3} + C_{xw}$</td>
<td>$C_{xp}$</td>
<td>$C_{xq}$</td>
<td>$C_{xr}$</td>
</tr>
<tr>
<td>2</td>
<td>$2 C_{y1} + C_{yu}$</td>
<td>$2 C_{y2} + C_{yv}$</td>
<td>$2 C_{y3} + C_{yw}$</td>
<td>$C_{yp}$</td>
<td>$C_{yq}$</td>
<td>$C_{yr}$</td>
</tr>
<tr>
<td>3</td>
<td>$2 C_{z1} + C_{zu}$</td>
<td>$2 C_{z2} + C_{zv}$</td>
<td>$2 C_{z3} + C_{zw}$</td>
<td>$C_{zp}$</td>
<td>$C_{zq}$</td>
<td>$C_{zr}$</td>
</tr>
<tr>
<td>4</td>
<td>$2 C_{p1} + C_{lu}$</td>
<td>$2 C_{p2} + C_{lv}$</td>
<td>$2 C_{p3} + C_{lw}$</td>
<td>$C_{lp}$</td>
<td>$C_{lq}$</td>
<td>$C_{lr}$</td>
</tr>
<tr>
<td>5</td>
<td>$2 C_{m1} + C_{mu}$</td>
<td>$2 C_{m2} + C_{mv}$</td>
<td>$2 C_{m3} + C_{mw}$</td>
<td>$C_{mp}$</td>
<td>$C_{mq}$</td>
<td>$C_{mr}$</td>
</tr>
<tr>
<td>6</td>
<td>$2 C_{n1} + C_{nu}$</td>
<td>$2 C_{n2} + C_{nv}$</td>
<td>$2 C_{n3} + C_{nv}$</td>
<td>$C_{np}$</td>
<td>$C_{nq}$</td>
<td>$C_{nr}$</td>
</tr>
</tbody>
</table>

For typical, symmetric aircraft configurations, it is often assumed, that changes of 'symmetric' forces (X, Z, M) depend only on the perturbations of symmetric velocities (u, w, q), and changes of 'asymmetric' forces depend on the perturbations of unsymmetric' velocities (v, l, p). Considering this, the table of computed aerodynamic derivatives has the form:
Tab. 3. List of aerodynamic derivatives computed with typical symmetric aircraft configurations

<table>
<thead>
<tr>
<th>i \ j</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(2C_{X_1}f_1 + C_{X_0})</td>
<td>(2C_{X_1}f_1 + C_{X_0})</td>
<td>(C_{X_1})</td>
<td>(C_{X_2})</td>
<td>(C_{X_3})</td>
<td>(C_{X_4})</td>
</tr>
<tr>
<td>2</td>
<td>(2C_{X_2}f_2 + C_{X_2})</td>
<td></td>
<td>(C_{X_2})</td>
<td>(C_{X_3})</td>
<td>(C_{X_4})</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>(2C_{X_3}f_3 + C_{X_3})</td>
<td></td>
<td></td>
<td>(C_{X_3})</td>
<td>(C_{X_4})</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>(2C_{X_4}f_4 + C_{X_4})</td>
<td></td>
<td></td>
<td></td>
<td>(C_{X_4})</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>(2C_{X_5}f_5 + C_{X_5})</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>(C_{X_5})</td>
</tr>
<tr>
<td>6</td>
<td>(2C_{X_6}f_6 + C_{X_6})</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>(C_{X_6})</td>
</tr>
</tbody>
</table>

The derivatives of aerodynamic coefficients were computed as quotients of finite differences, for example:

\[
C_{Z_0} = \frac{\Delta C_z}{V_A}
\]

\[
\Delta C_z = C_{Z(\alpha + \Delta\alpha)} - C_{Z(\alpha)} \approx \frac{\Delta\alpha}{V_A} \quad (\Delta\alpha \text{ in radians}).
\]

\[
C_{q_0} = \frac{\Delta C_z}{q \cdot b}
\]

where \(b\) is wing span.

For low values of Mach number \((M<0.5)\) the changes of the aerodynamic coefficients with the flight speed are very low, and the second elements of derivative definitions in the first column of Table 6.2 \((c_{Xp}, c_{Zp}, c_{Mp})\) are zeros.

4.2. Results for the cruise configuration

The non-dimensional aerodynamic derivatives were computed for \(\alpha = 0^\circ\) and \(\Delta\alpha = \Delta\beta = 1^\circ\). \(S_{ref} = 21.0\ m^2\), \(c_{ref} = 1.568\ m\), \(b_{ref} = 13.823\ m\).

The increments of angular velocity \(p = q = r = 1\ rad/s\). The values of aerodynamic derivatives are listed in the Tables 4 and 5.

Tab. 4. Non-dimensional stability derivatives computed for steady-level flight in cruise configuration

<table>
<thead>
<tr>
<th>i \ j</th>
<th>1 ((p_X))</th>
<th>2 ((p_Y))</th>
<th>3 ((p_Z))</th>
<th>4 ((p_L))</th>
<th>5 ((p_M))</th>
<th>6 ((p_N))</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-0.05154</td>
<td>-0.95771</td>
<td>-0.84459</td>
<td>-0.16808</td>
<td>-2.21801</td>
<td>0.20815</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

TRANSACTIONS OF THE INSTITUTE OF AVIATION No 207
4.3. Force and moment derivatives with respect to deflections of control surfaces

The derivatives with respect to deflections of control surfaces (horizontal tail and rudder) were determined as quotients of force and moment coefficient increments obtained for configurations with surface deflected at a small angle (2°–5°). Values of these derivatives are shown in Table 6.4.

The forces and moments were computed on the total aircraft wetted surface and the coefficients are referenced to $S_{ref}$, $c_{ref}$, $b_{ref}$ of the aircraft.

Control surface deflection was modelled through the deforming of the control-surface area of the mesh, without creating a slot between the two elements (Fig. 6.17).

This method was applied for the modelling of the deflection of elevator and rudder.

The simplified approach gives adequate results for total forces and moments acting on aircraft, but does not allow for the determination of hinge moments, as it does not account for flow in slots with regions of recirculation. Hinge moments of ailerons, elevator or rudder can be determined experimentally, or by field methods of CFD.
5. CONCLUSIONS

The CODA3Dpanel3dbl code, developed in IoA, was applied within the CESAR project for calculations of aerodynamic characteristic of baseline aircrafts. The code is a practical realisation of panel method coupled with boundary layer analysis using viscous-inviscid interaction methodology.

The aerodynamic characteristics determined using the CODA3Dpanel3dbl code are in agreement with both flight-test data and results of wind-tunnel experiments. The most visible difference is the over-prediction of the $c_{L_{\text{max}}}$ and $\alpha_{cL_{\text{max}}}$ by the present method. This difference may be an effect of not taking into account several factors influencing flow in the boundary layer, as fully three-dimensional and unsteady form of strong separation on the wing, influence of propeller wake or surface roughness. Another reason for this over-prediction may be dependence of the adopted model of boundary layer flow on parameters obtained in experimental investigations of boundary layer which have to be selected from certain range.

This introduces dependence of the solution on arbitrary decision of the software user or programmer. However, lower accuracy of the flow solution at high angles of attack also characterises CFD RANS solvers whose results depend strongly on the selected model of turbulence and grid quality.

Sometimes a three-dimensional grid of sufficiently high quality for resolving viscous flow around a complete aircraft configuration may require too much computer resources to be applicable by a small design company.

On the other hand, there is another possibility of coupling panel method with experimental results, which allows for determination of nonlinear characteristics at high angles of attack. This approach modifies circulation in wing strips such that local two-dimensional relation between circulation and angle of attack agrees with experimental airfoil characteristics [6]. This approach is fast and low-cost with regard to computer resources and may serve as indication for the precision of the characteristics obtained using viscous-inviscid interaction approach.
APPLICATION OF A PANEL METHOD WITH VISCOUS-INVISCID INTERACTION FOR THE DETERMINATION...
PRACE INSTYTUTU LOTNICTWA
spis treści czterech poprzednich zeszytów

Zeszyt 1/2010 (203)
M. Jeż, et al.: POLISH INSTITUTE OF AVIATION IN THE EUROPEAN RESEARCH AREA

Zeszyt 2/2010 (204)
G. Rarata: FENOMEN GÓRNEJ GRANICY WYBUCHOWOŚCI (GGW) PALIW W POWIETRZU I TLENIE W WARUNKACH
PODWYŻSZONEGO CIŚNIENIA I TEMPERATURY

Zeszyt 3/2010 (205)
A. Baron, M. Maczka, K. Piwek: THE CHALLENGE OF MOBILITY IN EUROPE
J. Rohacs: EVALUATION OF THE AIR TRANSPORT EFFICIENCY DEFINITIONS AND THEIR IMPACT ON THE EUROPEAN
PERSONAL AIR TRANSPORTATION SYSTEM DEVELOPMENT
I. Laplace, C. Chertier, A. Baron, M. Maczka: MOBILITY IN EUROPEAN COUNTRIES
I. Laplace, C. Chertier, A. Baron, M. Maczka: SYNTHESIS OF EPATS DEMAND ESTIMATION
A. Baron: EPATS AIRCRAFT MISSIONS SPECIFICATION
A. Baron: INTELLIGENT PERSONALIZED AIR TRANSPORTATION SYSTEM (IPATS)
A. Baron: AIR TRANSPORT EFFICIENCY AND ITS MEASURES
K. Piwek, A. Iwaniuk, W. Gnarowski: THE EUROPEAN PERSONAL AIR TRANSPORTATION SYSTEM (EPATS) STUDY
PROJECT: A SYSTEMATIC APPROACH TO A PERSONAL AIR TRANSPORT BASED ON SMALL AIRCRAFT AND SMALL
LOCAL AIRPORTS IN EUROPE

Zeszyt 4/2010 (206)
P. Boguszewicz, P. Wodyński: ODPORNOŚĆ SZYB CZOŁOWYCH SZYBKICH POJAZDÓW SZYNOWYCH NA ZDERZENIA Z
TZW.CIAŁAMI OBČYMI
W. Dąbrowski, S. Popowski: ESTYMACJA AZYMUTU Z WYKORZYSTANIEM POMIARÓW INERCJALNYCH
A. Gałach: CYKL TWORZENIA OPROGRAMOWANIA NA PRZYKŁADZIE PROJEKTU SOFIA
J. Graffstein, M. Krawczyk: SIEĆ TELEMETRYCZNĄ DO BADAŃ BEZDECHU
LOTNICZych
M. Jeż: EKOLOGICZNE PROBLEMY PORTU LOTNICZEGO
S. Kachel, A. Kozakiewicz: METODY PROGNOZOWANIA ROZWOJU OBIEKTU NA ETAPIE PROJEKTOWANIA STATKÓW
POWIETRZNYCH
M. Krawczycy, T. Rogalski: FUNKCJE REKONFIGURACJI LOTU- PRÓBA POPRAWY BEZPIECZEŃSTWA LOTU W PRZYPADKU
DZIÀAŁÓW O CHARAKTERZE TERRORYSTYCZNYM
M. Krawczycy, T. Rogalski: GŁÓWNEZAŁÓŻENIA ORAZ PRZEBIEG BADAŃ EKSPERYMENTALNEGO SYSTEMU STEROWANIA
SAMOLOTEM I-23 MANAGER
B. Naumienko, G. Rarata: AKTUALNE POTRZEBY I MOŻLIWOŚCI WYTWARZANIA PALIWA TYPU BIO-JET
M. Parafiniak: ANALIZA STEROWANIA WRNIEKIM ŚMIgłowCA BEZPILOTOWEGO
Z. Pagowski, K. Snopkiewicz: COOPAIR-LA- WYTYCZNE DLA WSPÓŁPRACY BADAŃcwIEJ MIĘDZY KRAJAMI UNII EURO-
PEJSKIEJ I AMERYKI ŁACIŃSKIEJ W LOTNICTWIE I TRANSPORcie LOTNICZYCH
M. Pokorski: Życie ZASOBÓW W CYKLU Życia KONSTRUKCJI: SAMOLOT, SAMOCHÓD
G. Rarata: INNOWACJAYNA TECHNOLOGIA USUWANIA NO2 Z GAZÓW ODLOTOWYCH ELEKTROWI ORAZ ELEKTRO-
CIEPŁOWNYCH ZA POMOCĄ HTP
W. Zdrojewski: BADANIA EKSPERYMENTALNE MODELU TURBiny WODnej TYPu MICHELL-BANKI