Czech Technical University in Prague, Faculty of Mechanical Engineering

Verified Unsteady Model for Analysis of Contra-Rotating Propeller Aerodynamics

FACULTY OF MECHANICAL ENGINEERING CTU IN PRAGUE

Ing. Vít Štorch

Supervisor: **prof. Ing. Jiří Nožička, CSc.** ***Co-supervisor:*** **Ing. Jan Čížek, Ph.D.**

Doctoral thesis in the field of Thermodynamics and Fluid Dynamics

Motivation and current state of the art

Contra-rotating propellers (CRP) are two propellers rotating about the same axis of rotation in opposite directions. Small size contra-rotating propellers are becoming popular as efficient propulsion units with low or negligible reaction torque for unmanned aerial vehicles. Wider spread is prevented by limited analysis tools, high generated noise and high performance expectations that are often not being fulfilled. Blade element momentum theory, lifting line method and actuator disc approach are among the methods used either separately, or combined into a more capable computational tools. None of these methods takes into account the unsteady viscous nature of the flow together with the real three-dimensional geometry of the propeller blades. Some research has been also conducted using CFD finite volume methods, however the computational time is preventing the use of these methods for large scale analysis and tasks such as multi-parametric optimization.

Goals and chosen approach

The aim of the thesis is to develop a computational model capable of detailed analysis of contra-rotating propellers subject to low Reynolds number flow accounting for various aspect of the flow ignored by other researchers. The aim is also to describe the properties of contra-rotating propellers using such advanced computational model and answer important questions regarding CRP system performance. The formulation of three main objectives is based on the review of literature and is given as follows:

1. Creating a viscous-inviscid interaction model that allows coupling of an advanced integral boundary model to a 3D panel method. This model must be fast enough to maintain the important advantage over CFD codes - speed of solution.
2. Creating an unsteady force-free wake model compatible with contra-rotating propeller configuration which would handle blade-wake interactions and allow accurate resolving of instantaneous wake shapes and induced velocity fields.
3. Describing properties of a contra-rotating propeller system under low Reynolds number flow regimes, especially: • Fluctuation of forces and torques during revolution; • Influence of propeller distance; • Sensitivity to the angle of free stream flow; • Comparison of a CRP system to an equivalent single propeller; • Influence of the ratio of rotational speeds of both propellers.

Computational model

3D panel method

Solves Laplace's equation:

$$\nabla^2\Phi = \frac{\partial^2\Phi}{\partial x^2} + \frac{\partial^2\Phi}{\partial y^2} + \frac{\partial^2\Phi}{\partial z^2} = 0 \quad (1)$$

With boundary condition on the body surface:

$$\vec{n} \cdot \vec{c} = 0 \quad (2)$$

By suitable superposition of flat source quadrilateral panels with source strength σ and vortex ring panels with circulation Γ, zero normal flow boundary condition is satisfied at collocation points at the center of each surface panel. This is performed by solving a system of N equations for N unknown circulations:

$$\vec{n}_i \cdot \left[\vec{c}_{i,rel} + \vec{c}_{i,ind} + \sum_j^N \vec{b}_{ij}\Gamma_j + \sum_j^N \vec{a}_{ij}\sigma_j\right] = 0 \quad (3)$$

Force-free vortex wake

It is formed by vortex ring panels. Every time step, new panel row is formed behind the trailing edge with circulation determined from the Kutta condition. The wake is convected downstream in the direction of local velocity so no forces are acting on the simulated wake sheet surface. Vortex filaments in the vortex wake behave according to the Helmholtz's theorems.

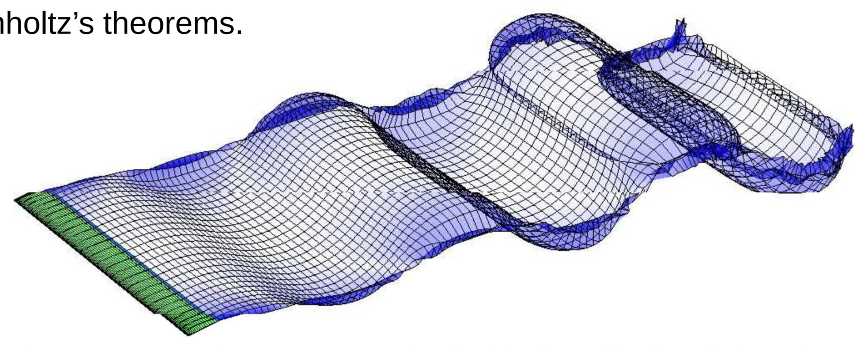

Fig. 1 – Force-free vortex wake behind oscillating finite wing

2D boundary layer model

Integral form two-equation boundary layer model similar to the one used in XFOIL panel code by M. Drela is used. The viscous layer in inviscid model is simulated by displacement of the surface in normal direction by the displacement thickness δ*. The governing equations of boundary layer model are the momentum equation and kinetic energy shape parameter equation (Eq. (4) and (5)):

$$\frac{d\theta}{d\xi} + (2+H)\frac{\theta}{u_e}\frac{du_e}{d\xi} = \frac{C_f}{2} \quad (4)$$

$$\theta\frac{dH^*}{d\xi} + H^*(1-H)\frac{\theta}{u_e}\frac{du_e}{d\xi} = 2C_D - H^*\frac{C_f}{8} \quad (5)$$

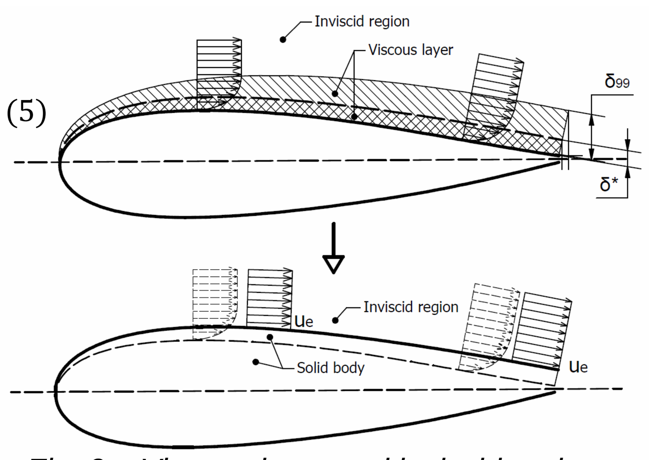

Fig. 2 – Viscous layer and inviscid region

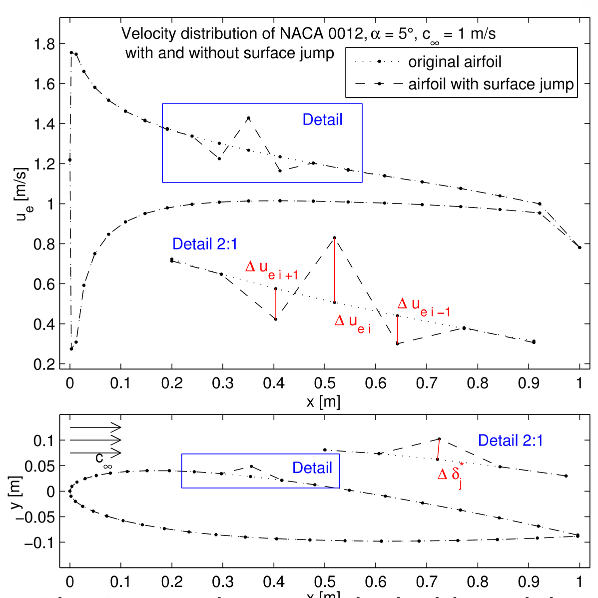

Fig. 3 – Replacement inviscid model – response of surface velocity to surface displacement in normal direction

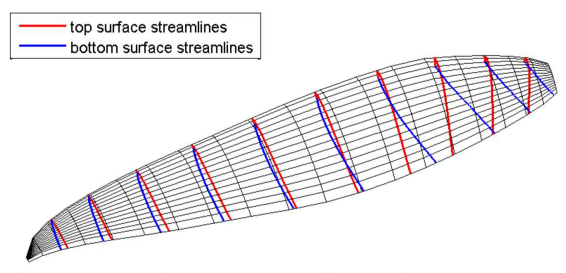

Fig. 4 – Surface streamlines for 2D boundary layer calculation

Equations (4) and (5) are completed by turbulent and laminar closure equations and together with auxiliary equation for shear stress coefficient form a system which is solved by a Newton iteration method in a downstream marching algorithm. Transition is detected by the e^n method.

New method of coupling the 2D boundary layer to inviscid solver has been developed. The interaction law which relates edge velocity u_e and displacement thickness takes form:

$$u_{e\,i,NEW} = u_{e\,i,OLD} + d_{ii}\left(\delta^*_{i,NEW} - \delta^*_{i,OLD}\right) \quad (6), \qquad d_{ii} = \frac{du_{e\,i}}{d\delta^*_i} \approx \frac{2u_e}{(\xi_i - \xi_{i-1})} \quad (7)$$

The interaction law forms basis of portable boundary layer model which only requires the inviscid solution to be calculated once. Edge velocity updates between boundary layer passes are calculated as follows (see also Fig. 3):

$$u_{e\,i} = u_{e\,i,orig} + d_{ii}(\delta^*_i - 0.5\delta^*_{i-1} - 0.5\delta^*_{i+1}) \quad (8)$$

Experimental verification

Experimental work was performed on a 4-component aerodynamic scale built for this purpose. Measured quantities included a pair of torques and thrusts of each propeller and respective rotational speeds.

Fig. 6 – Experimental setup

Propellers were driven by a coaxial drive composed of a pair of electric motors. This way the inflow path is clear of any obstructions.

One issue that this configuration brings is axial friction between both coaxial shafts which affects the measured distribution of thrusts between the shafts. The total thrust and torque of the system were however unaffected, which results in good match of the global performance parameters with numerical results.

The setup allows setting two propeller distances and independent control of rotational speed of each propeller. The propellers were measured under static condition i.e. with zero free stream velocity.

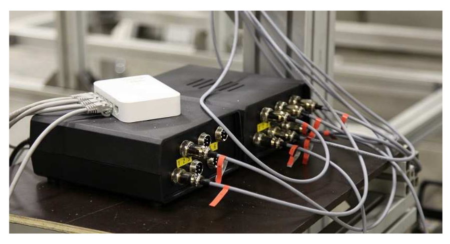

Fig. 7 – A/D converters

The results of measurement of performance parameters are presented together with numerical results

Supplementary measurement of noise was performed together with performance parameters to provide additional information important for assessment of a CRP system.

Fig. 8 – Pair of measured propellers

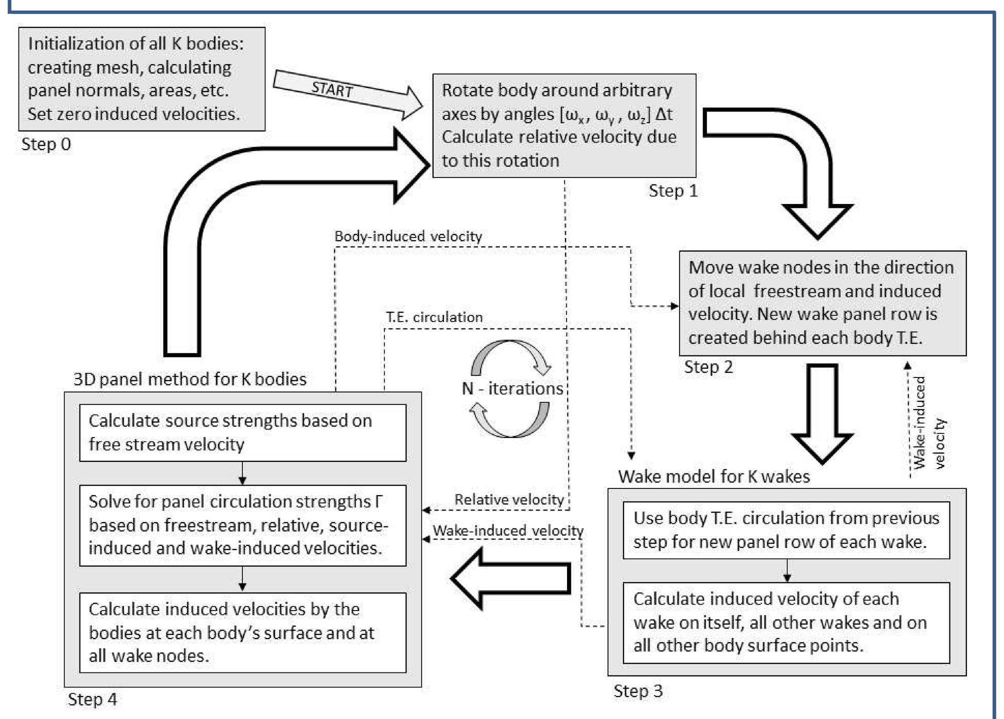

Fig. 5 – Block diagram of the unsteady solver

Results and analysis

Verification of computational methods was performed using results of single propeller in a wind tunnel and using results of static measurement of a CRP system. As an example, Figure of Merit calculated by 3D panel method with boundary layer and experimental data are compared in Fig. 9. The match between numerical and experimental data shows the importance of calculating boundary layer, especially when the blades are highly loaded and some separation is present.

The unsteady character of the forces and torques is apparent from Fig. 10 which illustrates the fluctuations of thrust and power of each propeller.

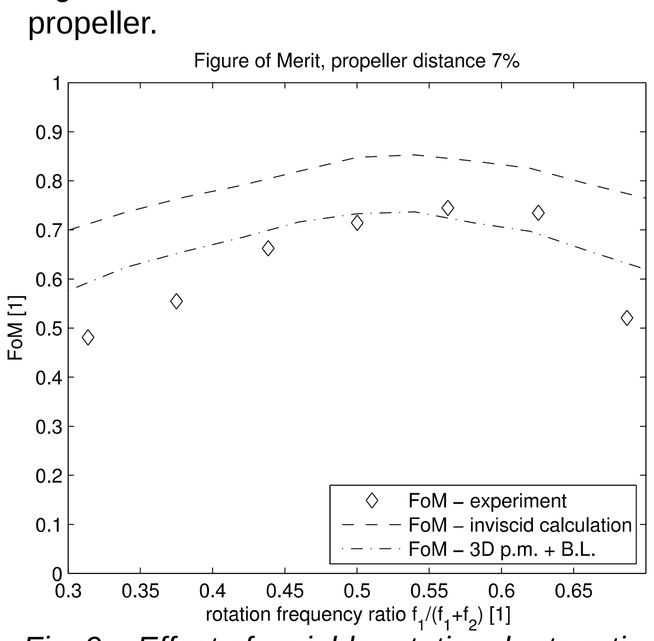

Fig. 9 – Effect of variable rotational rate ratio on Figure of Merit.

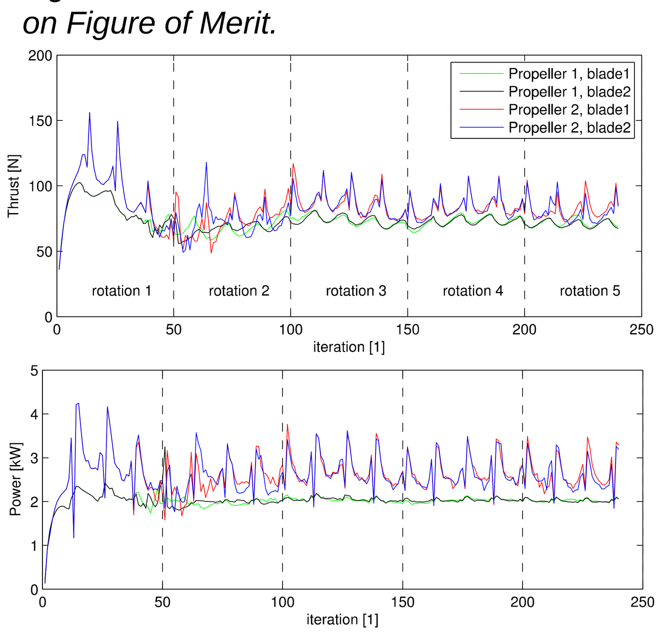

Fig. 10 – Example of fluctuations of thrust and power

Analysis of performance sensitivity to propeller distance revealed thrust redistribution and increase of Figure of Merit with increasing propeller distance.

Performance and wake shapes were studied for different advance ratios (Fig. 11) and performance curves were compared with an equivalent single propeller. Difference between wakes behind a pair of two-bladed contra-rotating propellers and single four-bladed propeller is in Fig. 12.

The capabilities of the method were well tested by the calculated test case with off-axis free stream velocity. The angle of incoming flow was varied between 0° and 60° and the effect on performance were analyzed. Wake shape for inflow angle 60° is in Fig. 13.

Definition of Figure of Merit used in the analysis:

$$FoM = \frac{(T_1 + T_2)^{3/2}}{(P_1 + P_2)\sqrt{2\rho\max(A_1, A_2)}} \quad (9)$$

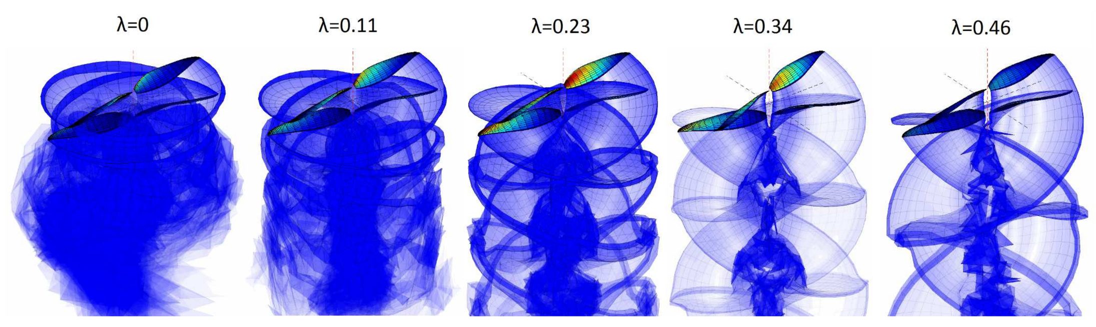

Fig. 11 – Wake shape development under different advance ratios

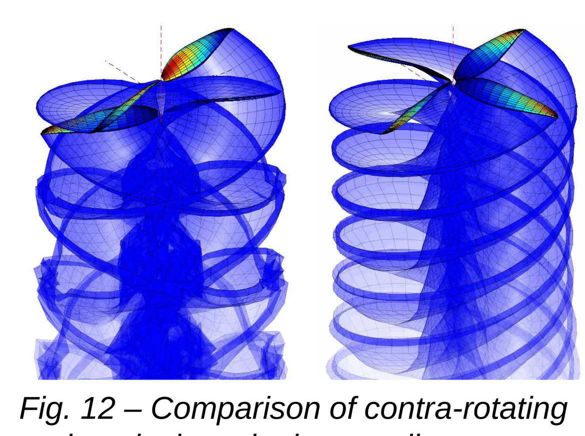

Fig. 12 – Comparison of contra-rotating and equivalent single propeller

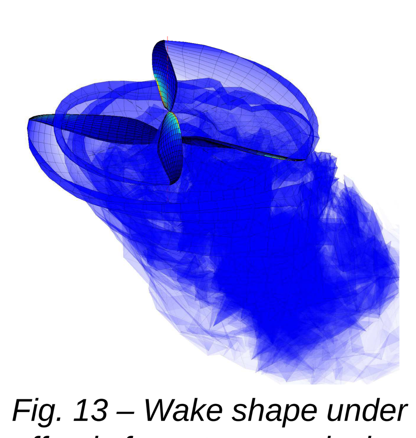

Fig. 13 – Wake shape under off-axis free stream velocity

Selected conclusions and recommendations

Coupling between inviscid 3D panel method and integral two equation 2D boundary layer model was performed using new interaction model. Portable boundary layer formulation uses linear interaction law and replacement inviscid model. Both the replacement inviscid model and interaction law benefit from fast method of estimation of interaction coefficients shown in Eq. (7). As a result, the calculation of boundary layer along selected streamlines on the surface of a rigid body proceeds without the need of 3D panel method solution between subsequent boundary layer passes. Results of experimental and numerical verifications on cases of CRP system and finite wing show promising results and suggest that this solution could be quite useful in wide ranges of low Reynolds numbers problems of flow past streamlined bodies. The method is especially useful in cases, where simpler methods such as lifting line or lifting surface methods fail to produce accurate solutions. These cases include low aspect ratio wings and blades, highly loaded rotors, blade and wing geometries with high sweep angle or rapid changes in planform shape or airfoil shape.

Several practical scenarios were studied and discussed. The following conclusions about the performance of CRP systems were reached:

- Response of a contra-rotating propeller system to change of ratio of frequencies of rotation showed that peak Figure of Merit is obtained for slightly different propeller rotational frequency ratio than 1:1. Although the most effective ratio depends on the exact geometry and conditions, it can be generalized that by controlling the ratio of rotation of propellers throughout the operation range, overall efficiency can be increased.
- Increasing propeller distance redistributes thrusts and slightly increases the values of Figure of Merit, while reducing noise. Propellers should be placed as far apart as possible and practical.
- Performance of a CRP system is initially insensitive to angle of off-axis free stream velocity, at higher angles of incidence the efficiency begins rising due to the effect of additional lift provided by forward flight component of velocity.
- CRP system and equivalent single propeller of the same diameter and blade solidity must be compared strictly at the same thrust level, otherwise incorrect conclusions may be drawn. At the same thrust levels, contra-rotating propeller system provides 1÷6% increase in efficiency over equivalent single propeller. Both upstream and downstream propellers are subject to fluctuating thrust force. The upstream propeller experiences rather smooth and gradual changes of thrust, while the downstream propeller is subject to sharp peaks in thrust and torque when the blades pass through wakes.

Relevant publications

[1] **V. Štorch**, J. Nožička, M. Brada, and J. Suchý. *Measurement of noise and its correlation to performance and geometry of small aircraft propellers.* In EFM 2015. EPJ Web of Conferences, 2016.

[2] J. Filipský and **V. Štorch**. *Comparison of propeller analysis methods and experimental data.* In Engineering Mechanics 2014, pp. 172-175. Brno University of Technology, 2014. ISSN 1805-8248.

[3] **V. Štorch** and J. Nožička. *3D panel methods for turbomachinery design.* In Fluid Mechanics and Thermodynamics, Proceedings of Students' Work in the Year 2013/2014. Czech Technical University in Prague, 2014.

[4] **V. Štorch**, J. Nožička, M. Brada, and J. Suchý. *Experimental verification of computational model for wind turbine blade geometry design.* In Experimental Fluid Mechanics 2014, pp. 572-577. Technical University of Liberec, 2014. EJP Web of Conferences.

[5] **V. Štorch** and J. Nožička. *A novel computational model for the analysis of contra-rotating propellers.* In Fluid Mechanics and Thermodynamics, Proceedings of Students' Work in the Year 2015/2016. Czech Technical University in Prague, 2016.

[6] **V. Štorch** and J. Nožička. *On viscous-inviscid interaction for boundary layer calculation using two-equation integral method.* In Fluid Mechanics and Thermodynamics, Proceedings of Students' Work in the Year 2014/2015. Czech Technical University in Prague, 2015.

[7] **V. Štorch** and J. Nožička. *Verification and possible applications of new 3d panel method with boundary layer.* In Fluid Mechanics and Thermodynamics, Proceedings of Students' Work in the Year 2016/2017. Czech Technical University in Prague, 2017.

[8] **V. Štorch**, J. Nožička, and M. Brada. *Experimental setup for measurement of contra-rotating propellers.* In Topical Problems of Fluid Mechanics 2017, pp. 285-294, 2017.

[9] **V. Štorch** and J. Nožička. *Contra-rotating propeller aerodynamics solved by a 3D panel method with coupled boundary layer.* Acta Polytechnica, Vol. 57, No.5, pp. 355-366, 2017.