29th Congress of the International Council
of the Aeronautical Sciences
St. Petersburg, Russia; September 7-12, 2014

THE JOINED WING SCALED DEMONSTRATOR RESULTS OF CFD

**Adam Dziubiński*, Sara Kuprianowicz*, Katarzyna Surmacz*,
Cezary Galiński*, Jerzy Żółtak***
***Instytut Lotnictwa**

Keywords: *computational aerodynamics, joined wing, simulation.*

Abstract

The following work is a short revision of CFD analysis of unconventional aircraft configuration, that has been done in The Institute of Aviation during the contribution within MOSUPS consortium. Following article contains short description of tools used in CFD simulation, a sketch of methodology, sample of results of calculations and explanation of tendencies and influences of certain phenomena appearing on flow field. In assumption it is an example, what kind of information can be assessed using CFD tools, whichare not always from top of the shell, but are fast and reliable in given flight conditions. This work was supported by The National Centre for Research and Development under grant No. PBS/A6/14/2012

1. General Introduction

The Joined Wing Concept has numerous advantages. There are two aerodynamic solutions that became crucial to create such configuration: the box wing and the staggered wings. The idea of box wing, proposed by Prandtl in 1924 [9], is based on assumption that using specific aerodynamic configuration of biplane, where upper wing and lower wing are connected at wingtip with plates, one can reduce significantly the induced drag of such aircraft. Using staggered wings, where one of the lifting surfaces in biplane configuration is moved forward, all benefits of aerodynamic interference: increased aircraft longitudinal stability, aerodynamic efficiency and maximum lift, can be also utilized. The idea of Joined Wing Concept is to move one of the wing in biplane configuration as much forward, as the horizontal stabilizer becomes unnecessary. Usually this is done that way, that one or both wings are swept. Then the induced drag is decreased by the wingtip plates.

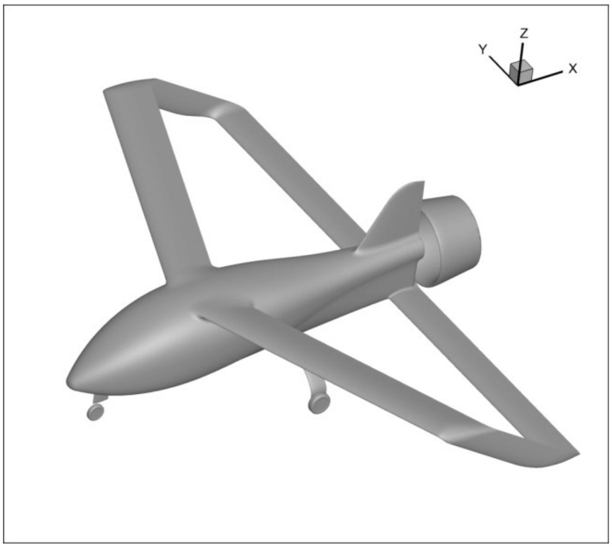


Fig. 1 The concept of joined wing demonstrator

That configuration has numerous non-aerodynamic advantages: stiffness of connected wings is increased so the aircraft can be lighter or can withstand more load than the similar without wingtip connection. The wingtip connection from structural point of view causes static indeterminacy, which can be good, but causes some technological issues: the tolerances in connections have to be tighter than in classic configuration. That was the reason, that such configuration hasn't been utilized in many designs. Another reason was aerodynamic complexity due to the close aerodynamic coupling [6]. Situation changes, when advanced computational tools become available.

Designers now have opportunity to use both CFD and FEM tools in process of multicriterial design. As new configuration is available, new aerodynamic issues and uncertainties appear, which also can be determined and solved using CFD tools [3]. The presented paper concerns about influence of selected design solutions on aerodynamic characteristics of joined wing configuration aircraft.

The Joined Wing Scaled Demonstrator Program (MOSUPS), is a consortium created to explore advantages of joined wing concept, and The Institute of Aviation is a main contributor. At actual stage, configuration of demonstrator is chosen as a joined wing with positive stagger (upper wing in front of the center of gravity, lower at the back). Front wing and wingtip plates are swept backward. The aircraft is designed in pusher configuration with ducted fan, and has a front-wheel type of landing gear. This configuration is an effect of research and experience achieved on previous designs created in Warsaw University of Technology [8].

2. Analysis

In CFD analysis done in The Institute of Aviation, a set of geometrical features has been considered to obtain their influence on aerodynamic characteristics. Since the wing configuration and surfaces has been defined by Warsaw University of Technology, our set has been defined as follows: airfoils and twist angles of both wings: front and rear, shape of a fuselage, fillets between fuselage and wings, shape of wingtip plates, influence of landing gear, influence of connection between rear wing, fuselage, and propeller duct. Shape of centerbody (engine cover including propeller spinner) and its influence on separation inside the propeller duct has also been considered. Almost all configurations have been tested to obtain working propeller influence. In general this information caused the designers to choose which modifications are necessary, and which are to be neglected in future works. Some elements, as for example the connection between duct wing and fuselage, has been chosen to be tested in wind tunnel and on flying model as one of the switchable modules. Usually those elements are worth considering, if their positive influence is not neglected by their mass. Also a lot of uncertainties can be understood and solved only after experimental test. Later on both CFD analyses results and wind tunnel tests are used to simulate behavior of an airplane as shown in [7] with methods presented in [4,5].

CFD simulations have been done using two tools: XFLR5 and ANSYS Fluent. A freeware XFLR5 software, an implementation of widely recognized XFOIL created by M. Drela [2] to analyze one-element airfoils, but its abilities have been extended to use lifting line theory and also vortex lattice and panel method to analyze wings and whole airframe configurations (Fig. 2) in static conditions and in obtaining stability data.

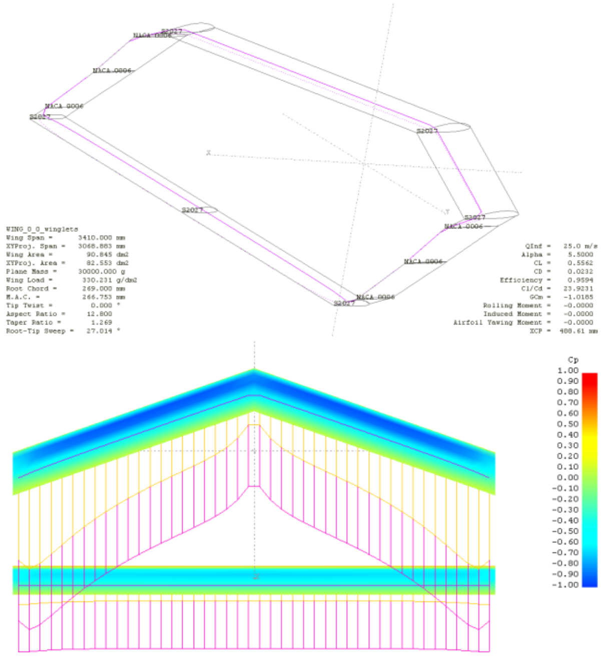


Fig. 2 XFLR5 screenshots (model and sample result) of twist angle analysis for simplified box wing configuration

XFLR5 to analyze a 2D airfoil uses a potential panel method with influence of boundary layer thickness. Turbulisation and separation are taken into account using secondary panel set, which are moved away from airfoil surface by the offset equal to boundary layer thickness in this area.

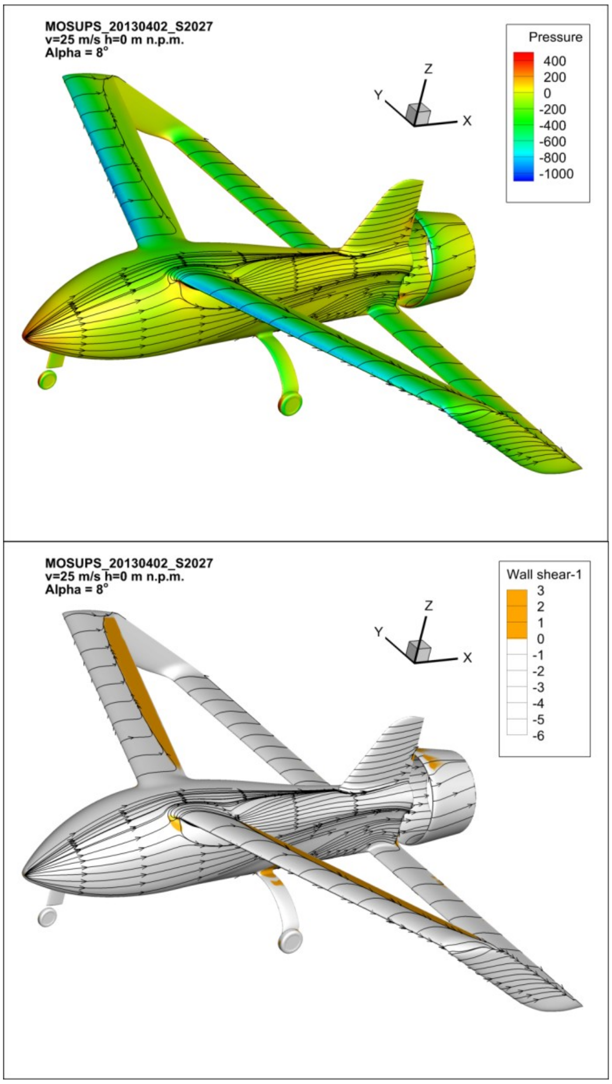


Fig. 3 Pressure and Wall shear distribution as visualization of detachment areas, composed with oil flow path lines visualization, obtained using ANSYS Fluent.

This way maximum lift, separation, and laminar and turbulent flows ranges (drag bucket phenomena) are well resolved. It is worth to remind, that on RC size objects (for example 0.1 - 0.2 m of chord), a laminar separation can occur, so those objects have to sustain different and sometimes more dangerous flight conditions than regular "general aviation" aircrafts. This software has been tested against experimental data and proved its credibility in analyzing object size of RC models [1], which is also the size of our demonstrator.

For detailed CFD solution a commercial ANSYS Fluent system, which utilizes the Finite Volume Method to obtain a RANS (Reynolds Averaged Navier Stokes equations) solution of flow field. has been used (Fig. 3).

2.1 Airfoil analysis

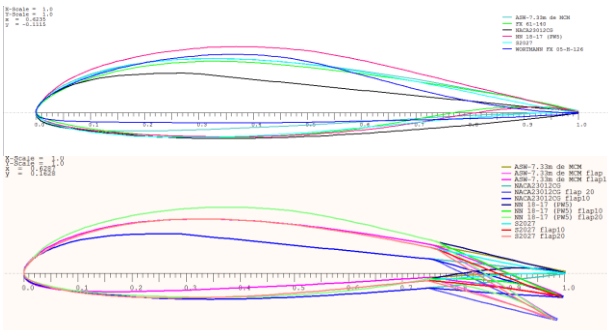


Fig. 4 Airfoil shapes comparison, clean and with flaps deflected.

a)

b)

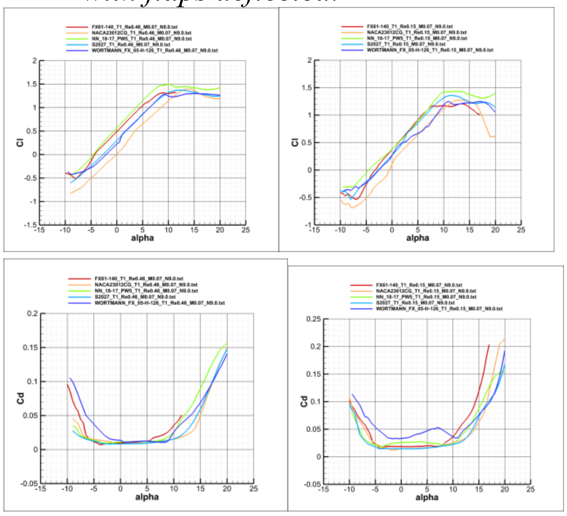


Fig. 5 Lift (a) and drag (b) characteristics for chosen clean airfoils with Reynolds number according to front and aft wing chord.

Airfoil characteristics have been obtained using XFLR5. Baseline airfoil NACA23012CG with flattened top surface has been compared to airfoils used in low speed aircraft design. Out of this set of airfoils, the S2027 has been chosen as the best suitable, because of its stable and linear characteristics, thickness, and low decrease of abilities (increase of drag, Fig. 6) when equipped with flap. Using in-build option in XFLR to modify airfoil shape in order to include flaps, a set of deformed airfoils has been made and tested (Fig. 4).

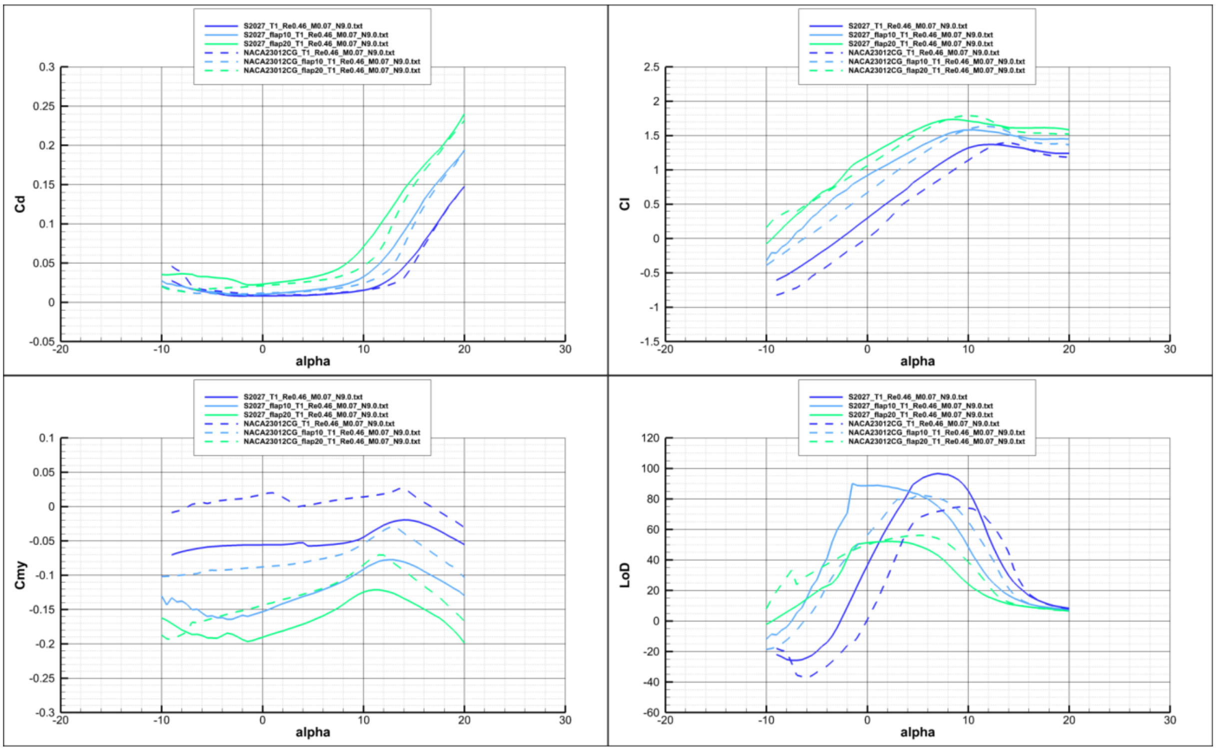


Fig. 6 Comparison of draglift moment and lift over drag characteristics between baseline and chosen airfoil for three different flap deflections.

a) *b)* *c)*

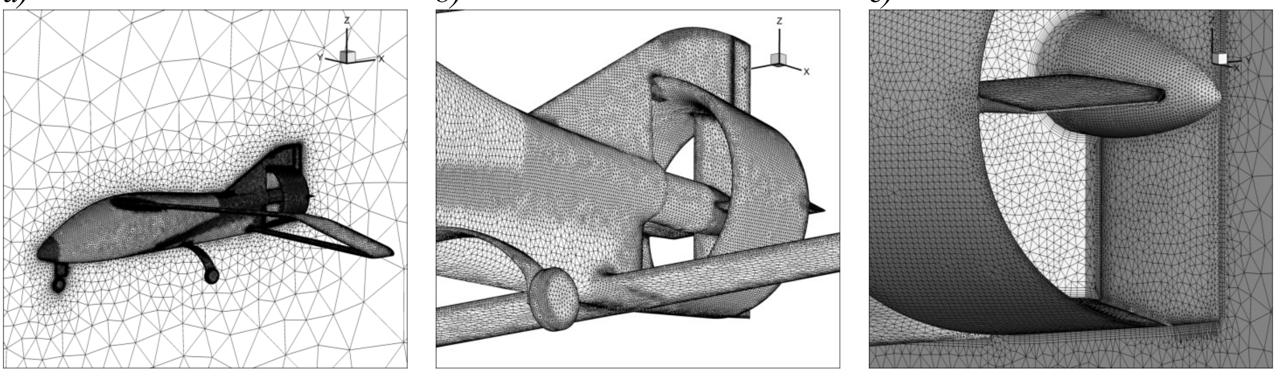
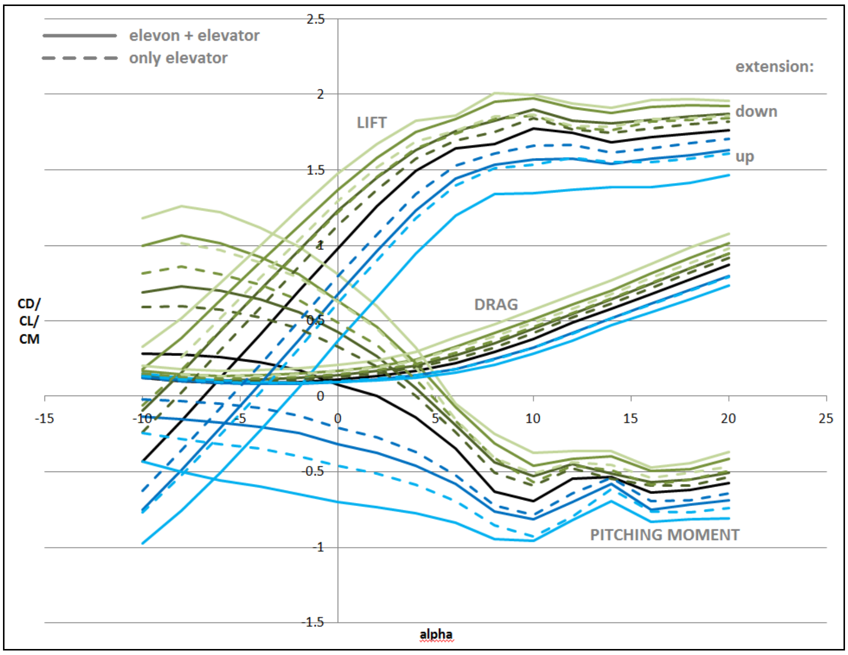


Fig. 7 a) size of mesh around an airplane, b)density distribution of mesh around ducted fan and c) ducted fan with surface of symmetry and actuator disc. On both surfaces the size of boundary layer mesh is visible.



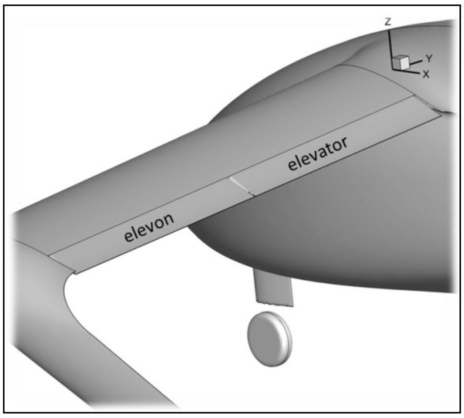


Fig. 1 Comparison between longitudal aerodynamic characteristics for different extension of the elevon and elevator.

2.2 Control surfaces effectiveness

In order to verify the control surfaces effectiveness, comercial RANS CFD solver using Spalart - Allmaras turbulence model has been used as it is usually recommended for external flow cases. 1.5 million elements tetrahedral grid (fig. 7a) has been created to model half of the aircraft (symmetry of geometry and flowfield has been assumed) . Around the model a few layers of hexahedral mesh have been created to properly obtain the influence of a boundary layer in terms of separation (fig. 7c).

The propeller influence has been modeled with actuator disc (constant pressure jump surface). All configurations have been tested in range of angle of attack between -10° and 20°, for assumed cruise velocity of 25 m/s. The model has been divided into functional parts (for example: fuselage, landing gear, front wing). The results obtained for whole range of angle of attack, could be presented for chosen elements or features as divided into parts. It provides useful information on aerodynamic interference of selected parts on each other, and ranges of angles of attack where this influence occurs.

In fig. 8 the qualitative results for different extension of elevator and influence of equal extension of elevon has been shown. On fig.9 and 10 respectively the separation areas and pressure distribution over the wings has been shown. The separation areas are simply distributions of shear stress component in flightwise direction. If the shear force is negative, that means flow is against the flight velocity, so the separation most probably occurred. This method of course fails, when the negative flow appears from the other reasons than the flow detachment. For example at stagnation area at the high angle of attack, when the part of air near nose of airfoil flows against the flight direction. But one could easily filter out those areas knowing where to find them.

For each combination of extension a different mesh has been created (Fig. 10). This approach has been used because of best quality of mesh and little if any complication for those cases in comparison to moving/deflecting mesh approach.

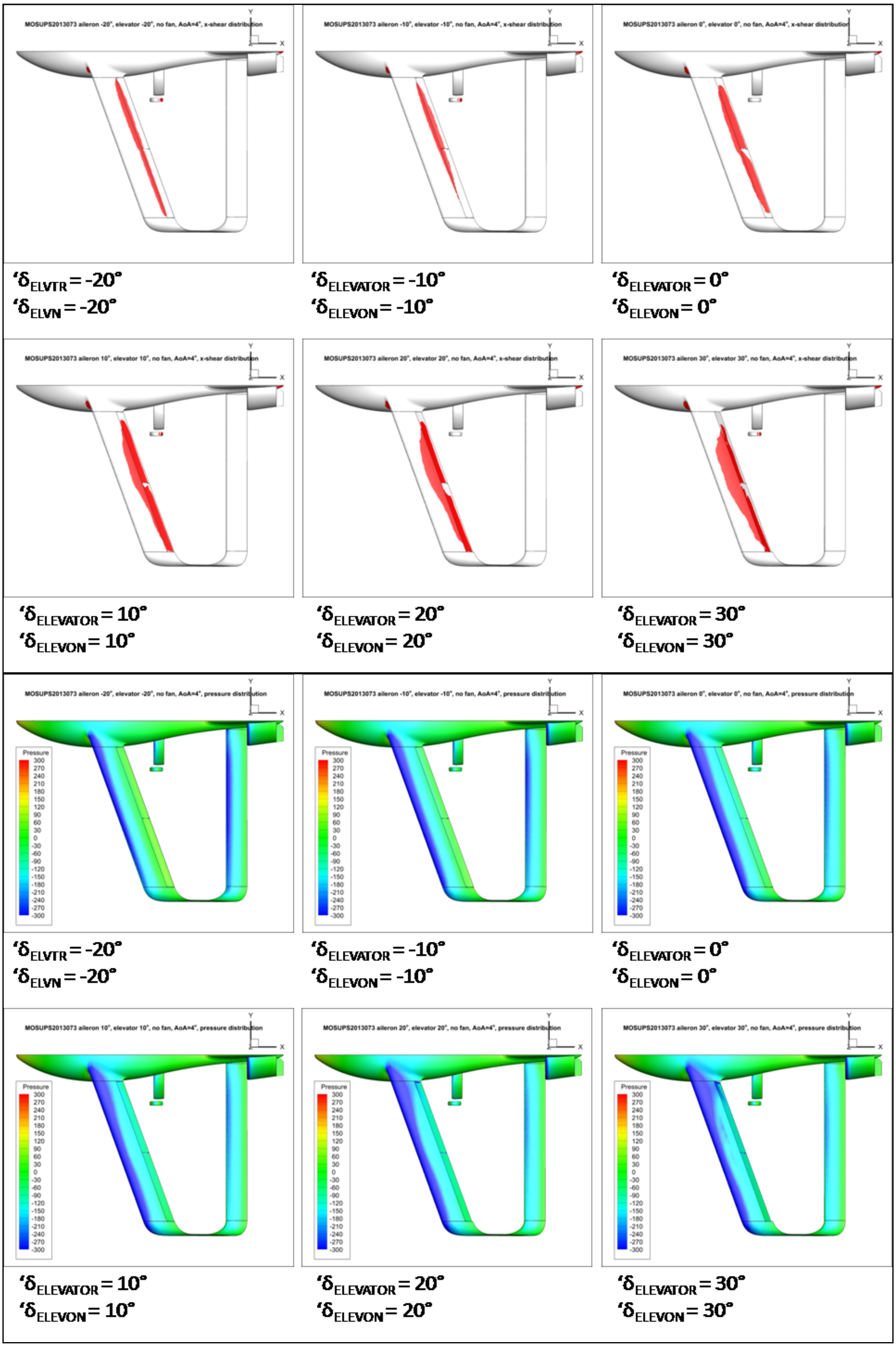


Fig. 10 Separation areas and pressure distribution for both elevon and elevator extended equally.

2.3 CFD analysis of small parts of geometry with uncertain aerodynamic influence.

Below the analysis of widely used modifications of geometry commonly trusted as improving aerodynamic characteristics, have been tested in object size of RC model. Those are the centerbodies for ducted fans, fillets, conical trailing shapes. Some of them proved to be useful in so slowly flying aircrafts, some not.

Baseline **Afterbody** **Afterbody+fillet**

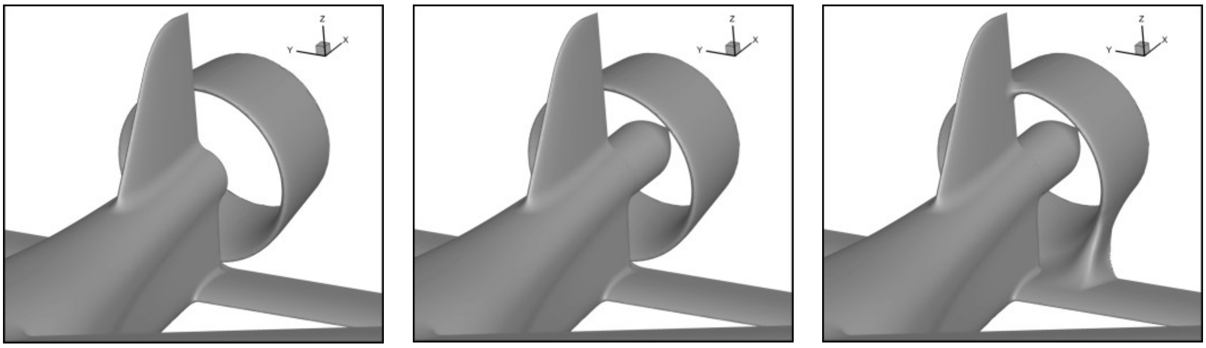


Fig. 11 Comparison of three analyzed configurations of tail

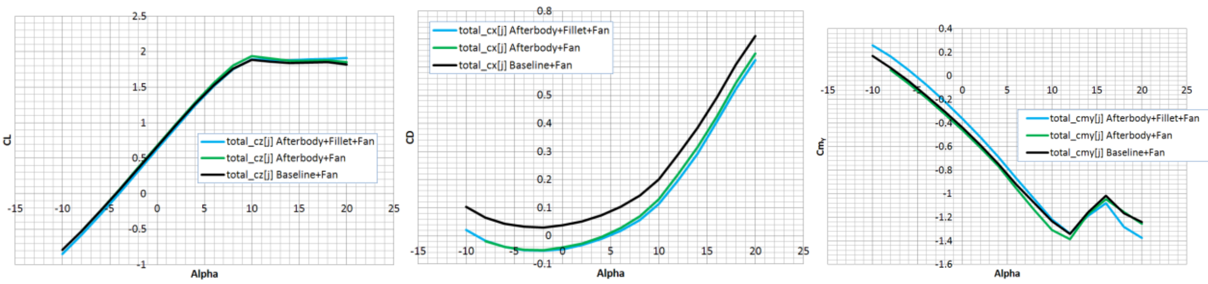


Fig. 12 Lift, drag and pitching moment characteristics for different modifications of tail

In fig 11 and 12 the cases and results of influence of center body and filleted wing to duct transition have been shown. The results are proving, that the center body has greater influence on drag of aircraft than filleted shape of duct.

Knowing the importance of centerbody, a shape of it has been also tested and results have been shown in fig.13. Even without conical shape the centerbody decreases the drag of aircraft in a non-negligible way ("afterbody_cut" case).

Another improvement is the conical segment. Influence of filleted mounting points of duct is so low, that it can be neglected. The negative values of drag are an effect of overpressure region caused by the actuator disc simulating the working propeller. For each case, even without centerbody and with different area of disc, the pressure jump on propeller have been set to create the same thrust.

Other interesting case, for which the results obtained with different than RANS methods are hardly (if ever) possible to get, is the analysis of separation inside fan duct with influence of working (still simulated as actuator disc) propeller, shown in fig 14. These results helped in creating a proper shape of engine nacelle and defining areas of possible cooling openings.

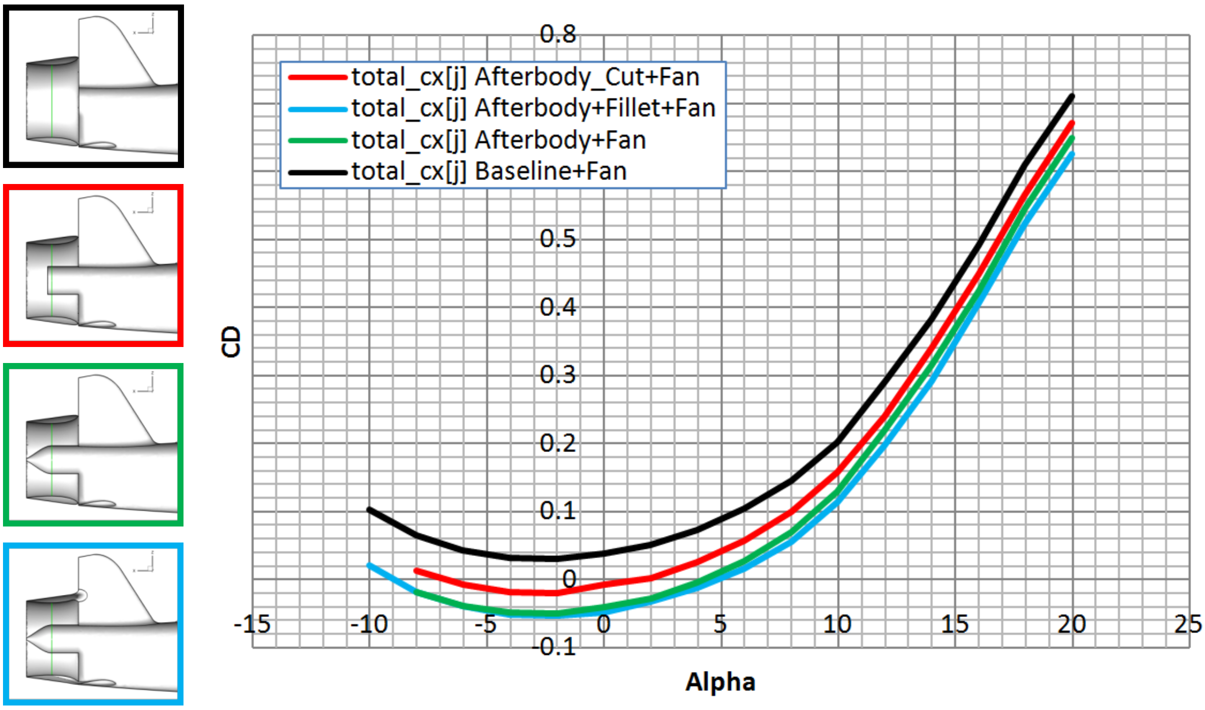


Fig. 13 Center body shape and fillets influence on aerodynamic drag of whole aircraft.

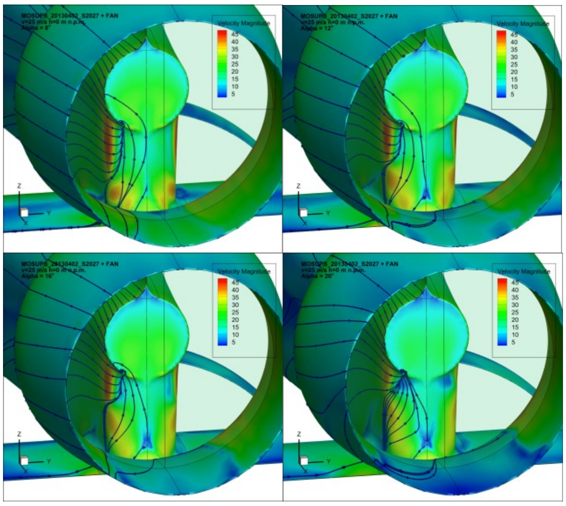


Fig. 14 Increase of separation area size on internal surface of the duct and rear surface of the engine nacelle with increase of angle of attack. Velocity magnitude distribution with pathlines.

3. Summary.

CFD again proved its functionality with obtaining crucial information about small design changes and its influence on aerodynamics without building an expensive wind tunnel model with many switchable modules. Great advantage is the ability to introduce advanced phenomena modeling at early stage of design (for example the propeller influence), so the design can become cheaper and more robust.

4. Acknowledgements.

This work was supported by The National Centre for Research and Development under grant No. PBS/A6/14/2012"

5. References.

[1] Deperrois A, *About XFLR5 calculations and experimental measurements* Rev1.1 (2009), www.xflr5.com

[2] Drela M, XFOIL: An Analysis and Design System for Low Reynolds Number Airfoils, *Low Reynolds Number Aerodynamics Proceedings of the Conference*, Notre Dame, Indiana, USA, 1989.

[3] Galiński C, Hajduk J, Kalinowski M, Wichulski M, Stefanek Ł, Inverted Joined Wing Scaled Demonstrator Programme, *Proceedings of the ICAS'2014 conference*, ICAS, Petersburg, Russia, 2014, ICAS2014-1.10ST, in print

[4] Goetzendorf-Grabowski T, Mieszalski D, Marcinkiewicz E.; Stability analysis using SDSA tool, *Progress in Aerospace Sciences,* Vol: 47 Issue: 8 Special Issue: SI pp. 636-646 Published: NOV 2011

[5] Goraj Z, Flight dynamics models used in different national and international projects *Aircraft Engineering and Aerospace Technology*, Volume 86, issue 3, 2014, pp.166-178

[6] Goraj Z, Kulicki P, Lasek M, Aircraft Stability Analysis For Strongly Coupled Aerodynamic Configuration, *Journal of Theoretical and Applied Mechanics*, Vol. 35, No. 1, 1997, pp.137-158.
[7] Lis M, Dziubiński A, Galiński C, Krysztofiak G, Ruchała P, Surmacz K, Predicted Flight Characteristics of the Inverted Joined Wing Scaled Demonstrator, *Proceedings of the ICAS'2014 conference,* ICAS, Petersburg, Russia, 2014, ICAS2014-P2.4.13, in print
[8] Mamla P, Galinski C, Basic Induced Drag Study of the Joined-Wing Aircraft, *Journal of Aircraft*, Vol. 46, No. 4, 2009, pp. 1438-1440.
[9] Prandtl L, Induced drag of multiplanes *NACA TN 182*, 1924.

6. Contact Author Email Address

mailto: adziubinski@ilot.edu.pl

Copyright Statement