Abstract: The range of unmanned aerial aircraft (UAV) is developing continuously generating new constructive solutions from small size and mass to those, which are comparable to piloted aircraft. The main reason of UAV use is due to lower cost of design, realization and operation (on flight hour) in comparison with human piloted aircraft. This paper presents the CFD analysis on a UAV tailless, after a 3D scanned real model.

Keywords: CFD, UAV, flying wing, aerodynamic parameters

1. INTRODUCTION

The range of unmanned aerial aircraft (UAV) is developing continuously generating new constructive solutions from small size and mass to those, which are comparable to piloted aircraft. The main reason of UAV use is due to lower cost of design, realization and operation (on flight hour) in comparison with human piloted aircraft. According to references [1, 2, 3, 4] UAV development included the flying wing (tailless) operating lonely or in network, see Fig. 1, [13, 14].

Constructive solutions of flying wing UAV are characterized by a series of requirements and exploitation limits on whole cycle of conception, fabrication and testing. UAV requirements and exploitation limits can be grouped on the following
categories: design concepts, aero-mechanical, materials and technologies, testing methods, flight safety and security and economical [5, 6, 7, 15 and 16].

2. CFD STUDIES FOR FLYING WING UAV

2.1. Preprocessing of model

The mock-up geometry was rebuilt with Ansys DesignModeler and Catia starting from scanned digital model (STL format) see Fig. 2, [8].

![FIG. 2 CFD model of flying wing](image1)

The mesh was generated with Numeca/Hexpress and has 3.8 millions of hexahedral cells. The grid is refined to properly capture the leading and trailing edge curvature the junction between wing and vertical empennage, see Figs. 3 and 4, [11, 12].

![FIG. 3 Computational domain and mesh](image2)  ![FIG. 4 Mesh on flying wing surface](image3)

2.2. Computational Cases

The solver used in this study was Ansys Fluent [8]. In order to reduce the computational effort, a symmetry plane was introduced. The governing equations are the Euler system. Furthermore, the fluid is assumed incompressible because the infinite upstream velocity is low (30 m/s). To solve numerically the Euler system, we used the SIMPLE algorithm due to its robustness and convergence rate for incompressible flows [9, 10]. The computational cases are given in table 1.

<table>
<thead>
<tr>
<th>Velocity (V)</th>
<th>30 m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>Angle of attack $\alpha$ (°)</td>
<td>-6</td>
</tr>
</tbody>
</table>

The reference values used to obtain lift and drag coefficients and relative pressure are given in Table 2.
Table 2. Reference parameters

<table>
<thead>
<tr>
<th></th>
<th>p</th>
<th>101325 Pa</th>
<th>ρ</th>
<th>1,225 kg/m³</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>0.29 m²</td>
<td>V</td>
<td>30 m/s</td>
<td></td>
</tr>
</tbody>
</table>

2.3. Results and discussions

The CFD results are given in table 3 and in Figs. 6-9.

Table 3 CFD results

<table>
<thead>
<tr>
<th>V</th>
<th>30 m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>α</td>
<td>C_L</td>
</tr>
<tr>
<td>-6°</td>
<td>0.074</td>
</tr>
<tr>
<td>-4°</td>
<td>0.227</td>
</tr>
<tr>
<td>-2°</td>
<td>0.383</td>
</tr>
<tr>
<td>0°</td>
<td>0.538</td>
</tr>
<tr>
<td>3°</td>
<td>0.760</td>
</tr>
<tr>
<td>6°</td>
<td>0.944</td>
</tr>
<tr>
<td>9°</td>
<td>1.012</td>
</tr>
<tr>
<td>12°</td>
<td>1.007</td>
</tr>
<tr>
<td>15°</td>
<td>0.926</td>
</tr>
<tr>
<td>18°</td>
<td>0.901</td>
</tr>
</tbody>
</table>

FIG. 5 Computed aerodynamic parameters, a- lift coefficient, b- drag coefficient, c, d- lift to drag ratio (C_L/C_D)

According to table 3 and Fig. 5, the maximal value of lift coefficient is at angle of attack of 9°; significant increase of drag coefficient is observed from angles of attack greater than 6°.
The position of model in the scanned geometry was considered at angle of attack of 0°. One observes an incidence phase difference from the visualization of CAD model and from the comparison between CFD and wind tunnel results, see Fig. 6.

One observes that the maximal lift coefficient is achieved for the angle of attack in the range 12° to 13°, see Fig. 7a; a significant increase of drag coefficient appears at angles of attack higher that 10° and the maximal lift to drag ratio is at angle of attack of around 1°, see Fig. 7c. The pressure difference on the pressure side and suction side of wing gives the lift. This is clearly illustrated in Fig. 8 and Fig 9.
3. CONCLUSIONS AND NEW RESEARCH DIRECTIONS

This CFD analysis for a flying wing UAV shows that the aerodynamic optimization is necessary. Furthermore, the CFD results have to be validated with the experimental ones obtained in subsonic wind tunnel. The above analysis shows the aerodynamic characteristics for a flying wing UAV obtained through 3D scan and simplified geometry in junction zones (wing- vertical empennages).

For a better analysis, it is necessary a finer mesh, which is suitable for RANS computations. Moreover, it is necessary to increase the number of test cases for a velocity range from 0 to 35 m/s and an angle of attack range from -10° to 20°. The future CFD
analyses could include the flights with the gliding angle up to 10° and with different twist angles.

ACKNOWLEDGMENT

This work is supported by the Executive Agency for Higher Education, Research, Development and Innovation Funding (UEFISCDI) under MASIM project (PN-II-PT-PCCA-2013-4-1349).

REFERENCES

[16] Udroiu R., Applications of additive manufacturing technologies for aerodynamic tests, ACADEMIC JOURNAL OF MANUFACTURING ENGINEERING, ISSN 1583-7904, VOL. 8, ISSUE 3/2010