Numerical Analysis of Laminar to Turbulent Transition on the DU91-W2-250 airfoil

M. Raciti Castelli, G. Grandi and E. Benini

Abstract—This paper presents a study of laminar to turbulent transition on a profile specifically designed for wind turbine blades, the DU91-W2-250, which belongs to a class of wind turbine dedicated airfoils, developed by Delft University of Technology. A comparison between the experimental behavior of the airfoil studied at Delft wind tunnel and the numerical predictions of the commercial CFD solver ANSYS FLUENT® has been performed. The prediction capabilities of the Spalart-Allmaras turbulence model and of the γ-0 Transitional model have been tested. A sensitivity analysis of the numerical results to the spatial domain discretization has also been performed using four different computational grids, which have been created using the mesh generator GAMBIT®. The comparison between experimental measurements and CFD results have allowed to determine the importance of the numerical prediction of the laminar to turbulent transition, in order not to overestimate airfoil friction drag due to a fully turbulent-regime flow computation.

Keywords—CFD, wind turbine, DU91-W2-250, laminar to turbulent transition

I. INTRODUCTION AND BACKGROUND

With its 2020 goals to increase the share of renewable energy in the overall energy mix to 20% and to cut carbon emissions by 20%, the EU is leading the world in terms of renewable energy deployment, exports and promotion. Today Europe gets approximately 20% of its electricity from renewable energy sources, including 5.3% from wind energy. That share will increase up to 2020 when, under the terms of the EU’s renewable energy directive, which sets legally binding targets for renewable energy in Europe, 34% of the EU’s total electricity consumption will come from renewable energy sources, with wind energy accounting for 14% [1]. In this scenario, the continuous quest for clean energy appears to be connected with the development of the aerodynamics of actual wind turbines, in order to achieve a growth of their performances, both for the classical horizontal-axis (HAWT) and also the vertical-axis (VAWT) concepts [2].

For the past years, it was common practice to exist using airfoil families, like the well known NACA series, for the design of wind turbine blades, however the need of furthering wind turbine technologies has led to the quest for alternatives.

Fig.1 Comparison between the DU91-W2-250 airfoil and a 5-digit NACA airfoil

The airfoil analyzed in the present work is the DU91-W2-250, which belongs to a class of wind turbine dedicated airfoils developed by Delft University of Technology. At present, DU airfoils are being used by various wind turbine manufacturers worldwide, in many different rotor blades.

The design of the DU91-W2-250 airfoil followed wind tunnel tests on a 25% thick NACA airfoil from the 6-3-4xx series, linearly scaled from 21%. To compensate for the resulting loss in lift of the upper surface, a certain amount of lower surface aft loading was incorporated, giving DU91-W2-250 the typical S-shape of the pressure side. This airfoil, like other 25% thick airfoils, has very high peak lift coefficient in the smooth condition and presents an acceptable performance in the rough situation, differently from classical NACA airfoils. The main features of the mid span airfoil are a good maximum lift to drag ratio and a smooth stall behavior [3] [4].

Every flow causes pressure and friction on the body surface, which result in forces and moments acting on the body itself. Nowadays, thanks to advances in numerical methods and computing power, the investigation and solution of the flow field around an airfoil has become relatively simple. By performing CFD analysis on the DU91-W2-250, together with turbulence and transition modeling testing, the main purpose of the present work is to investigate its behavior, with particular attention to the laminar to turbulent transition phenomena.

Lombardi et al. [5] tested the capability of a classical RANS solver of predicting the friction drag over a NACA 0012 airfoil for 0 deg angle of attack and compared CFD results with the values given by a coupled potential/boundary-layer method. The analyzed range of Reynolds numbers varied from 300,000 to 9,000,000. As a result, being the local skin friction coefficient defined as:

$$c_f = \frac{\tau_w}{(\frac{1}{2} \rho \cdot c \cdot V_{\infty}^2)}$$

(1)
the relative integral over the whole airfoil length resulted overestimated by all turbulence models - even using highly refined grids - because of their inherent inability to predict the boundary layer transition.

Lian and Shyy [6] coupled a Navier-Stokes solver and a Reynolds-averaged two-equation closure to study laminar to turbulent transition for low Reynolds number flows around a SD7003 airfoil, obtaining good agreement between numerical turbulent transition and the simulation results using fully turbulent SS T model. The analysis showed that the transition model can capture the main characteristic is, however, the capability of foreseeing laminar to turbulent transition. In fact, classical turbulence models, although widely used to calculate the pressure loads acting on blade profiles, are unable to predict the laminar-turbulent transition, resulting in poor prediction of rotor performance, caused by the overestimation of airfoil friction drag due to a fully turbulent-regime flow computation [14], especially for high values of the tip speed ratio where, due to the low range of blade relative angles of attack, the skin friction contribution to overall airfoil drag is quite relevant [15].

II. MODEL GEOMETRY

The present work was performed applying both Spalart-Allmaras turbulence model and \( \gamma-\theta \) Transitional model. The first one is a relatively simple one-equation model that solves a modeled transport equation for the kinetic eddy viscosity; this model was designed specifically for aerospace applications involving wall-bounded flows and was shown to give good results for boundary layers subjected to adverse pressure gradients [13].

The second one is based on the coupling of the SST transport equations with two other transport equations, one for the intermittency and one for the transition onset criteria, in terms of momentum-thickness Reynolds number [7] [8]. Its main characteristic is, however, the capability of foreseeing laminar to turbulent transition. In fact, classical turbulence models, although widely used to calculate the pressure loads acting on blade profiles, are unable to predict the laminar-turbulent transition, resulting in poor prediction of rotor performance, caused by the overestimation of airfoil friction drag due to a fully turbulent-regime flow computation [14], especially for high values of the tip speed ratio where, due to the low range of blade relative angles of attack, the skin friction contribution to overall airfoil drag is quite relevant [15].

Table I summarizes the experimental results of the tests.

<table>
<thead>
<tr>
<th>Denomination</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Airfoil section</td>
<td>DU91-W2-250</td>
</tr>
<tr>
<td>( c ) [m]</td>
<td>0.6</td>
</tr>
<tr>
<td>( \text{Re} ) [-]</td>
<td>3.0\times10^6</td>
</tr>
<tr>
<td>( \alpha ) [deg]</td>
<td>0.49</td>
</tr>
</tbody>
</table>

CFD analysis was performed using both Spalart-Allmaras turbulence model and \( \gamma-\theta \) Transitional model. The first one is a relatively simple one-equation model that solves a modeled transport equation for the kinetic eddy viscosity; this model was designed specifically for aerospace applications involving wall-bounded flows and was shown to give good results for boundary layers subjected to adverse pressure gradients [13].

The second one is based on the coupling of the SST transport equations with two other transport equations, one for the intermittency and one for the transition onset criteria, in terms of momentum-thickness Reynolds number [7] [8]. Its main characteristic is, however, the capability of foreseeing laminar to turbulent transition. In fact, classical turbulence models, although widely used to calculate the pressure loads acting on blade profiles, are unable to predict the laminar-turbulent transition, resulting in poor prediction of rotor performance, caused by the overestimation of airfoil friction drag due to a fully turbulent-regime flow computation [14], especially for high values of the tip speed ratio where, due to the low range of blade relative angles of attack, the skin friction contribution to overall airfoil drag is quite relevant [15].
dependent dimensionless distance that quantifies the degree of wall layer resolution, in formulas:

\[ y^+ = \frac{\rho u^+ y}{\mu} \]  

(2)

Each grid was called as “Mod” followed by a number from 1 to 4. Tables II and III summarize the main features of the adopted grids, as well as the resulting \( y^+ \) peak values.

### TABLE II

**MAIN MESH FEATURES AND \( y^+ \) VALUES OBTAINED USING THE SPALART-ALLMARA TURBULENCE MODEL**

<table>
<thead>
<tr>
<th>Name</th>
<th>First row [mm]</th>
<th>Growth factor [-]</th>
<th>Rows [-]</th>
<th>Suction side peak ( y^+ ) value [-]</th>
<th>Pressure side peak ( y^+ ) value [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mod1</td>
<td>0.7</td>
<td>1.2</td>
<td>4</td>
<td>110</td>
<td>110</td>
</tr>
<tr>
<td>Mod2</td>
<td>0.05</td>
<td>1.2</td>
<td>15</td>
<td>7.5</td>
<td>7</td>
</tr>
<tr>
<td>Mod3</td>
<td>0.025</td>
<td>1.2</td>
<td>17</td>
<td>3.8</td>
<td>3.5</td>
</tr>
<tr>
<td>Mod4</td>
<td>0.0125</td>
<td>1.2</td>
<td>21</td>
<td>1.85</td>
<td>1.8</td>
</tr>
</tbody>
</table>

### TABLE III

**MAIN MESH FEATURES AND \( y^+ \) VALUES OBTAINED USING THE \( \Gamma-\Theta \) TRANSITIONAL MODEL MODEL**

<table>
<thead>
<tr>
<th>Name</th>
<th>First row [mm]</th>
<th>Growth factor [-]</th>
<th>Rows [-]</th>
<th>Suction side peak ( y^+ ) value [-]</th>
<th>Pressure side peak ( y^+ ) value [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mod1</td>
<td>0.7</td>
<td>1.2</td>
<td>4</td>
<td>110</td>
<td>110</td>
</tr>
<tr>
<td>Mod2</td>
<td>0.05</td>
<td>1.2</td>
<td>15</td>
<td>7.5</td>
<td>7</td>
</tr>
<tr>
<td>Mod3</td>
<td>0.025</td>
<td>1.2</td>
<td>17</td>
<td>3.8</td>
<td>3.5</td>
</tr>
<tr>
<td>Mod4</td>
<td>0.0125</td>
<td>1.2</td>
<td>21</td>
<td>1.85</td>
<td>1.8</td>
</tr>
</tbody>
</table>

### III. DESCRIPTION OF THE NUMERICAL FLOW FIELD

As already mentioned in the previous section, all the adopted grids presented common geometric features, except for the areas close to the airfoil. As the aim of the numerical simulations was to explore the 2D flow field close to a blade profile, the computational domain was discretized into two macro-areas:

- a rectangular outer zone, determining the overall calculation domain, with a circular opening centered at 25% of chord length, which was identified as **Wind Tunnel sub-grid**, fixed;
- a circular inner zone, which was identified as **Airfoil sub-grid**, were grid points were clustered in order to obtain an accurate mesh setup of both the wall boundary layer and the airfoil wake.

Fig. 2 shows the main dimensions and boundary conditions of the **Wind Tunnel sub-grid** area. The computational domain width was set to 20 blade chords. In order to allow a full development of the wake behind the airfoil, inlet and outlet boundary conditions were placed respectively 10 blade chord upwind and 20 blade chord downwind with respect to the airfoil test section.

Two **symmetry** boundary conditions were used for the two side walls. The boundary between **Wind Tunnel sub-grid** and **Airfoil sub-grid** was set as an **interior**, thus ensuring the continuity of the flow field.

---

Fig. 2 Main dimensions and boundary conditions of the **Wind Tunnel sub-grid** area

---

**TABLE IV**

**MAIN REFERENCE VALUES OF THE NUMERICAL FLOW FIELD**

<table>
<thead>
<tr>
<th>Denomination</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \rho ) [kg/m³]</td>
<td>1.225</td>
</tr>
<tr>
<td>( \mu ) [Pa·s]</td>
<td>1.7894·10⁻⁵</td>
</tr>
<tr>
<td>( V_x ) [m/s]</td>
<td>73</td>
</tr>
<tr>
<td>( V_y ) [m/s]</td>
<td>0</td>
</tr>
</tbody>
</table>

---

IV. DISCRETIZATION OF THE NUMERICAL FLOW FIELD

A totally unstructured mesh was chosen for the **Wind Tunnel sub-grid**, in order to reduce the engineering time to prepare the CFD simulations.
With regard to the Airfoil sub-grid, the computational grids around the tested airfoil were constructed from lower topologies to higher ones, adopting appropriate size functions, in order to cluster grid points near the leading edge and the trailing edge of the blade profile, so as to improve the CFD code capability of determining lift, drag and the laminar to turbulent transition onset. A high-quality structured mesh was created close to the airfoil surface, in order to better capture the boundary layer, while outside the boundary layer region a triangular unstructured grid was created using proper size functions. Figs. from 3 to 8 show the main features of the computational grids around the tested airfoil for the four candidate meshes. Some details of the grid close to the leading and trailing edge are also shown for Mod 3 mesh.

V. SIMULATED FLOW CONDITIONS

Simulations were performed using the commercial RANS solver ANSYS FLUENT®, which implements 2-D Reynolds-averaged Navier-Stokes equations using a finite volume-finite element based solver. A segregated solver, implicit formulation, was chosen for unsteady flow computation. The fluid was assumed to be incompressible. As a global convergence criterion, residuals were set to $10^{-5}$.

VI. RESULTS AND DISCUSSION

Table V shows the DU91-W2-250 reference lift and drag values, respectively defined as:

$$C_l = \frac{L}{\frac{1}{2} \rho c V_\infty^2}$$ (3)

$$C_d = \frac{D}{\frac{1}{2} \rho c V_\infty^2}$$ (4)

which were measured at Delft University Low-speed Wind Tunnel [12] for an angle of attack of $\alpha = 0.49$ deg.

<table>
<thead>
<tr>
<th>Denomination</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_l$ [-]</td>
<td>0.469</td>
</tr>
<tr>
<td>$C_d$ [-]</td>
<td>0.00766</td>
</tr>
</tbody>
</table>

### TABLE V
DU91-W2-250 REFERENCE LIFT AND DRAG VALUES MEASURED AT DELFT UNIVERSITY LOW-SPEED WIND TUNNEL FOR $\alpha = 0.49$ DEG (FROM: [12])

<table>
<thead>
<tr>
<th>Denomination</th>
<th>$C_l$ [-]</th>
<th>$C_d$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mod1</td>
<td>0.38176</td>
<td>0.01558 (+103.3%)</td>
</tr>
<tr>
<td>Mod2</td>
<td>0.42409</td>
<td>0.02199 (+97.5%)</td>
</tr>
<tr>
<td>Mod3</td>
<td>0.42691</td>
<td>0.01307 (+70.6%)</td>
</tr>
<tr>
<td>Mod4</td>
<td>0.42211</td>
<td>0.01331 (+73.8%)</td>
</tr>
</tbody>
</table>

### TABLE VI
NUMERICAL PREDICTED LIFT COEFFICIENT AND RELATIVE PERCENTAGE DEVIATION WITH RESPECT TO EXPERIMENTAL MEASUREMENTS, SPALART-ALLMARAS TURBULENCE MODEL
Tables VI and VII show a comparison between computed and measured lift and drag coefficients, as a function of the adopted spatial discretization, for both Spalart-Allmaras turbulence model and $\gamma$-$\theta$ Transitional model. It clearly appears that Mod 3 and Mod4 grids determine the best results in terms of percentage deviations from the experimental measurements. Moreover, the prediction capabilities of the $\gamma$-$\theta$ Transitional model appear to be quite higher with respect to the Spalart-Allmaras turbulence model, as far as drag prediction is concerned.

### TABLE VII

<table>
<thead>
<tr>
<th>Denomination</th>
<th>$C_1$ [-]</th>
<th>$C_2$ [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mod1</td>
<td>0.31888</td>
<td>0.01633</td>
</tr>
<tr>
<td></td>
<td>(-32%)</td>
<td>(+113.3%)</td>
</tr>
<tr>
<td>Mod2</td>
<td>0.3367</td>
<td>0.01409</td>
</tr>
<tr>
<td></td>
<td>(-28.3%)</td>
<td>(+83.9%)</td>
</tr>
<tr>
<td>Mod3</td>
<td>0.44596</td>
<td>0.00886</td>
</tr>
<tr>
<td></td>
<td>(-4.9%)</td>
<td>(+15.7%)</td>
</tr>
<tr>
<td>Mod4</td>
<td>0.44381</td>
<td>0.00908</td>
</tr>
<tr>
<td></td>
<td>(-5.4%)</td>
<td>(+18.5%)</td>
</tr>
</tbody>
</table>

The higher quality of Mod 3 grid spacing, confirmed by the good results obtained by the computation of the aerodynamic coefficients, clearly appears from the distribution of the relative $y^+$ parameter between the prescribed values $1 \leq y^+ \leq 5$. The use of the proposed statistical methodology for the analysis of the quality of the grid is independent from the tested airfoil geometry or angle of attack and could be generalized for the whole polar of the considered airfoil.
Further work should be performed, in order to numerically determine the whole range of lift and drag coefficients for the considered airfoil geometry, as a function of the angle of attack.

The registered drag overestimation (+70.6%) of the Spalart-Allmaras turbulence model is connected to the overprediction of the skin friction drag due to a fully turbulent simulation of the flow field close to the airfoil, being the area under the skin friction curve equal to the overall skin friction coefficient. The described phenomenon is quite relevant for the airfoil pressure side, as evidenced by the orange arrows in Fig. 18.

Finally, once Mod 3 mesh was identified as the better spatial discretization, the distribution of the skin friction coefficient along both airfoil pressure and suction sides was investigated. Figs. 17 and 18 show a comparison between the numerical predicted skin friction drag using both Spalart-Allmaras turbulence model and $\gamma-\theta$ Transitional model. As can be clearly seen, the Spalart-Allmaras turbulence model is unable to predict the laminar to turbulent transition, which is registered by the $\gamma-\theta$ Transitional model (evidenced by the green arrows) for nearly 35% of chord length.

VII. CONCLUSIONS AND FUTURE WORKS

Laminar to turbulent transition on the DU91-W2-250 airfoil was investigated by means of a CFD simulation of the flow field using the $\gamma-\theta$ Transitional model. Numerical results were compared to both wind tunnel experimental measurements and Spalart-Allmaras turbulence model predictions. A significant improvement in drag prediction was obtained by using the transitional computation (15.7% overestimation) in comparison with the fully turbulent simulation (70.6% overestimation). The analysis of the distribution of the skin friction coefficient along the airfoil pressure and suction sides confirmed the $\gamma-\theta$ Transitional model capability of foreseeing
the laminar to turbulent transition which, for the present case, was estimated at 35% of the chord length.

Finally, grid quality was investigated by means of a statistical methodology, obtained by subdividing the global $y'$ range in several sub-intervals and computing the probability of the $y'$ value at any given point along the airfoil to be comprised inside each sub-interval.

Further work should be performed, in order to extend the present analysis to a wider range of angles of attack.

**NOMENCLATURE**

- $c$ [m] chord length
- $C_d$ [-] drag coefficient
- $C_l$ [-] lift coefficient
- $c_f$ [-] skin friction coefficient
- $D$ [N] drag force acting on the airfoil
- $L$ [N] lift force acting on the airfoil
- $P$ [-] probability for the $y'$ value at any given point along the airfoil to be comprised inside each sub-interval
- $R_e$ [-] airfoil Reynolds number
- $u_τ$ [m/s] tangential wall velocity
- $V_x$ [m/s] free-stream wind velocity, x-component
- $V_y$ [m/s] free-stream wind velocity, y-component
- $V_∞$ [m/s] free-stream wind velocity
- $x$ [m] curvilinear coordinate along airfoil suction/pressure side
- $X$ [-] number of grid elements having the $y'$ comprised in the considered interval
- $y$ [m] wall-grid centroid distance
- $y'$ [-] wall y-plus
- $Y$ [-] total number of grid elements on airfoil pressure/suction side
- $α$ [deg] airfoil angle of attack
- $µ$ [Pa·s] dynamic viscosity
- $ρ$ [kg/m$^3$] air density
- $τ_w$ [N/m$^2$] wall shear stress

**REFERENCES**


[12] W.A. Timmer, W. A. and R. P. J. O. M. van Rooij, *Afoil DU91-W2-250 Coordinates and Measurements in Delft University 1.25x1.80 m Low-speed Wind tunnel*, data provided by direct contact from the authors.

