A NEW MATHEMATICAL MODEL FOR HIGH THICKNESS COANDA EFFECT WALL JETS

Valeriu DRĂGAN

Politehnica University of Bucharest, Romania

Abstract: *The paper presents a new mathematical model for calculating the pressure coefficient distribution across a Coandã ramp. Theoretical as well as computational models are used in order to obtain a good correlation between the proposed model and the other, more labor intensive methods. Banner's theoretical model is used as a baseline and two blending functions are defined and calculated for corrections necessary near the blowing slot and the separation point. An empirical method was used to determine the angular position of the separation point in good correlation with the CFD data. Numerical simulations were carried out for a simple case which was then used to validate the proposed CEPA model. Although more experimental data should be acquired in order to increase the level of accuracy of the model, the paper shows that the proposed model fairly predicts the pressure distribution for the cases studied.*

Keywords: *Coandã effect, wall jet, RANS, CFD, CEPA*

1. INTRODUCTION

The Coandã effect is almost ubicuos to aeronautical applications, especially in the recent applications which incorporate fluidic high lift devices (Guo et al. 2011) Fixed wing aircraft are not the only applications, no tail rotorcraft also benefit from the use of this effect (Cîrciu et al. 2010).

As seen in the literature, most applications rely on fluidic pelicular jets blowing tangentially to a curve surface to either create lift directly (Drăgan^{a} 2012) or indirectly, acting as a high lift device shown in Fig.1 (Drăgan^{b} 2012).

Due to the ease of use of modern computational fluid dynamics methods, theoretical development of the Coandã effect was all but abandoned in recent years.

Therefore, the hereby paper seeks to extend a theoretical model and establish a frame for which it would be useful as a pre-design development tool.

Certain semi-empirical models exist in the state of the art however they are generally developed from experimental results in which the h/R ratio is close to 1% and are therefore inaccurate at higher h/R ratios. The model sought in this paper refers to higher h/R ratios, since Upper Surface Blown wing designs impose it.

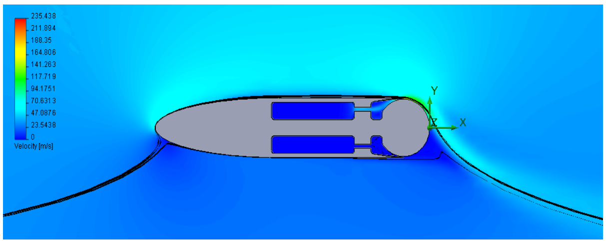

Fig. 1. Entrainment lift airfoil using the Coanda effect (Drăgan^{b} 2012)

2. THE THEORETICAL MODEL

2.1 Banner's pressure coefficient calculation

The early attempts to mathematically describe the Coandã effect relied upon the balance between the pressure and centrifugal forces exerted on an infinitesimal control volume, Fig.2. One of the first theories that dealt with the calculation of the average pressure coefficient across a circular ramp is given by (Banner, 1964). His demonstration considers a small volume of the Coandã flow of mass dm. For h/R ratios smaller than one, the balance between the pressure forces and the centrifugal forces acting upon the volume can be expressed by equalizing the two equations:

$$F_c = \frac{\rho R d\theta dR u^2}{R} \tag{1}$$

$$F_p = R d\theta dP \tag{2}$$

Where dθ is the infinitesimal angular element.

The pressure drop along the jet is then expressed by

$$\Delta P = P_{static\ jet} - P_{atm} = -\frac{\rho v^2 h}{R} = -\frac{2h}{R}\cdot\frac{\rho v^2}{2} \tag{3}$$

by introducing the thrust of an element of thickness h

$$T_H = \rho v^2 h \tag{4}$$

we can write

$$\Delta P = -\frac{T_H}{R} \tag{5}$$

therefore, by defining the pressure coefficient

$$C_p = \frac{\Delta P}{\frac{\rho v^2}{2}} \tag{6}$$

we reach the expression

$$C_p = -\frac{2h}{R} \tag{7}$$

Another equivalent demonstration is made for thicker jets (where h is comparable with the curvature radius R) by (Roderick 1961).

$$C_p = -\frac{2h}{R}\left(1 + \frac{h}{2R}\right) \tag{8}$$

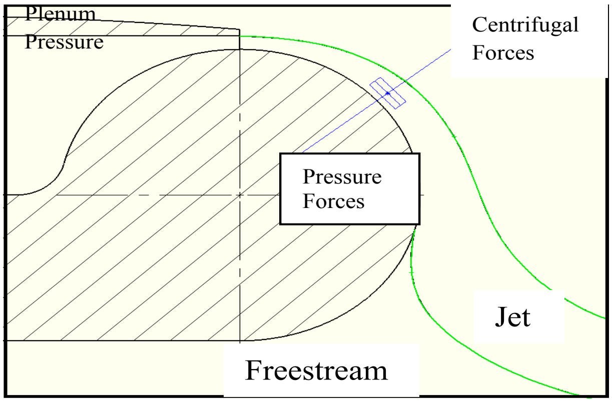

Fig. 2 The basic setup for the current case

The models presented however are crude and do not fully describe the physical problem. This is one of the reasons for which more sophisticated models, such as (Lewinsky and Yeh, 1989) and Saeed's semi-empirical model (Saeed, 2011), were created.

Although these models are designed to compute the velocity flow field around the Coandã surface they have many limitations, i.e. Saeed's model only works for small curvature radii (h/R <<1).

An alternative, semi-empirical model, is presented in (Dragan^c, 2012) however, due to the availability of the experimental data, it too is limited to h/R ratios smaller than or equal to ten.

The advantage of the Banner and Roderick models is that they work well especially for high h/R ratios. This is because the total pressure losses across the ramp are less important than in the low h/R ratio cases where the flow rapidly loses the total pressure. In order to confirm this, the hereby paper presents a series of computational fluid dynamics tests (CFD).

2.2 The Computational Fluid Dynamics comparison.

As shown by (Bakker, 2005), the conventional RANS models based on the concept of turbulent viscosity are unable to predict accurate flow separation points due to the exacerbated turbulent production. Numerical comparative studies (Frunzulică et al. 2011) also shows that the curvature corrections brought to the two equation turbulence models only mitigate the problem, without providing a physically correct model for Coandă flows. Therefore a different, more physically sound, RANS model was used. The five equation Reynolds Stress Model (RSM) is regarded as the best RANS viscosity model. It has also been confirmed as a tool for the numerical simulation of entrainment airfoils (Slomski et al. 2003), such as the one presented in Fig.1.

A simple ramp geometry was chosen for the 2D dp simulation which was discretized using a structured mapped mesh presented in Fig.3.

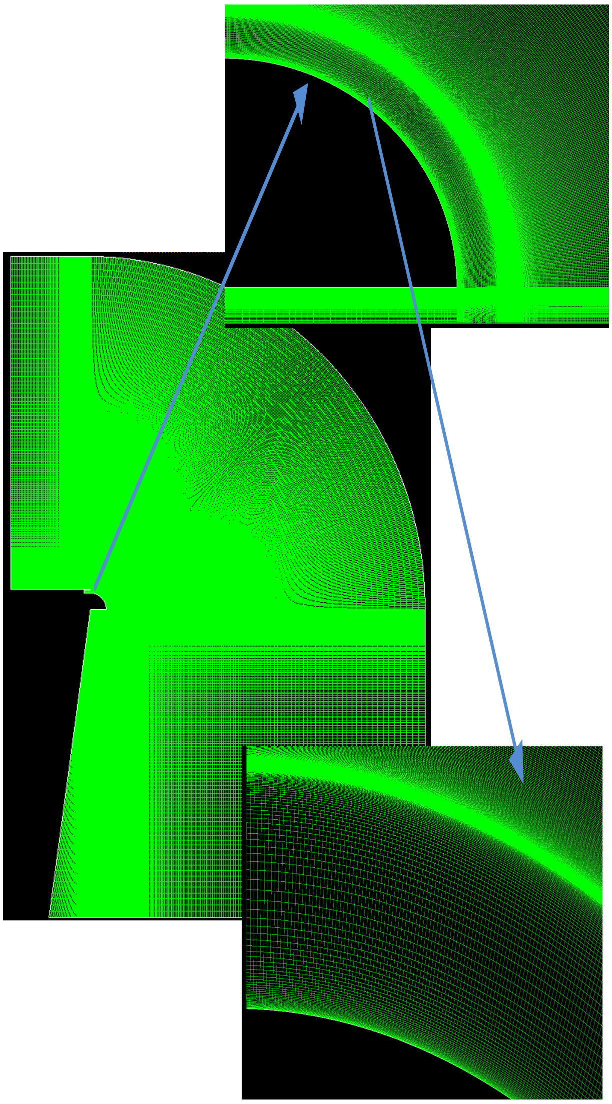

Fig.3 The computational mesh

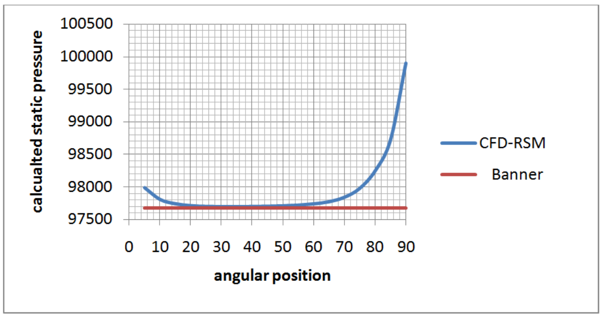

Fig. 4 The comparison between the basic Banner model and the CFD calculation performed with the Reynolds Stress Model

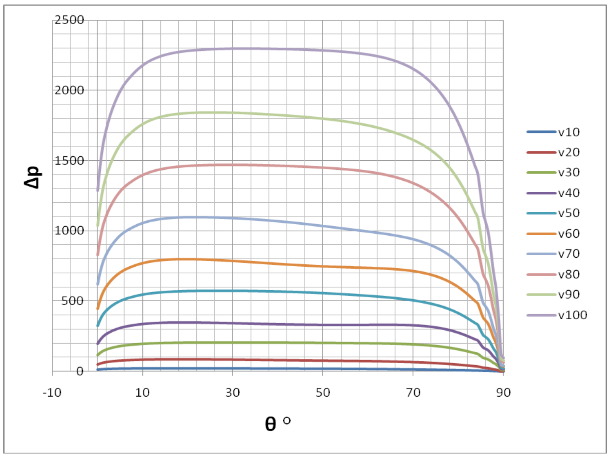

Fig.5 The pressure distribution across the ramp span for different blowing velocities

2.3 The proposed correction functions.

As seen, the Banner equation predicts quite well the pressure drop calculated by the CFD RSM method. This coupled with the fact that the pressure profile for all blowing velocities is quite flat, may be used to develop a new mathematical model.

A quick observation shows that the pressure estimates near the slot and near the separation point are not correctly calculated by Banner's model.

In order to improve on the model we must then define two correction functions, F_1 and F_2 to describe the pressure behavior in the problematic regions.

It is also worth mentioning that the velocity profile near the blowing slot is not calculated in either Lewinsky's or Saeed's models, therefore the attempt to describe it is unique to this proposed model.

In order to construct the F_1 function we must first determine the geometric parameter that influences it – so that the equation is relative to it rather than an arbitrary length or angular position (which would lead to accuracy issues).

After studying many geometric cases it was concluded that the domain of our function spans in the interval $0 < \theta < \frac{h}{R}\frac{180}{\pi}$.

This is also intuitive since the height of the slot influences the jet's boundary layer development, (Wygnanski, 2002).

By using non-linear curve fitting methods, we then determined the F_1 correction for near-slot pressure distribution

$$F_1 = 0.6014 + 0.4056 \cdot \left\{ 1 - \exp\left[-3.198 \left(\frac{\theta}{\frac{h}{R}\frac{180}{\pi}} \right) \right] \right\} \quad (9)$$

In a similar manner, we observe that the domain of the F_2 correction function may also be linked to a geometric parameter which can be calculated individually for each case. It appears that the interval for F_2 is

$$[\theta_{sep} - 0.143 \cdot \theta_{sep}; \theta_{sep} + 0.143 \cdot \theta_{sep}].$$

One way of calculating the angle of flow separation is by using the Sleeman-Phelps equation (Yen, 1982).

$$\theta_{sep} = 6.69 \left(\frac{R}{h} \right)^{1,54} \quad (10)$$

Therefore, using the same curve fitting techniques, a correction function was derived for the second problematic section of the Coandã flow:

$$F_2 = 1.02915 - 0.02915 \cdot \exp\left(\frac{\theta - \left(\theta_{sep} - 0,143 \cdot \theta_{sep}\right)}{(\theta_{sep} + 0,143 \cdot \theta_{sep}) - (\theta_{sep} - 0,143 \cdot \theta_{sep})} \right) \quad (11)$$

Figures 6 and 7 shows the correlation between the deduced correction functions and the results obtained by the CFD numerical simulations. Both equations are in good agreement with the CFD data.

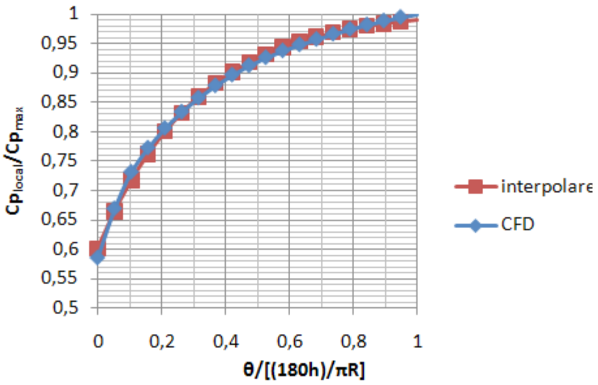

Fig. 6 The correlation between the F_1 correction function near the blowing slot and the CFD simulation results

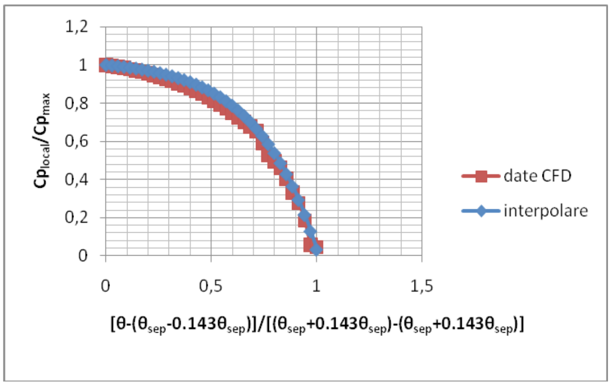

Fig. 7 The correlation between the F_2 correction function near the separation point and the CFD simulation results

As a measure for the accuracy of the proposed method we may define the Coandã effect lift efficiency:

$$\varsigma_C = \frac{L}{F} = \frac{R}{h} C_p \cdot \sin\left(\frac{\theta_{sep}}{2} \right) \quad (12)$$

where

L is the absolute value of the lift force generated by the Coandã effect

F is the thrust of the blowing jet used

Table 1. Comparison between the CEPA model and the RANS simulation results

Method used	θ_{sep}	η_C
CFD Reynolds Stress Model	~80°	119%
CEPA Proposed model	79.77°	120%

3. CONCLUSIONS

The paper presents an improved method for calculating the pressure coefficient near a Coandã ramp which is circulated by a thin wall jet. The proposed Coandã Effect Pressure Approximation (CEPA) model dwells on the theoretical frame given by Banner with the addition of two blending functions for the regions which have significant deviation from the theoretical model. Another addition to the model the calculation of the separation point by using Sleeman and Phelps's experimental data. A CFD model was also tested as a benchmark, having a h/R ratio of 20%. Due to the blending functions, the CEPA model based on Banner's theory coupled with the Sleeman-Phelps equation accurately described the pressure coefficient distribution across the ramp.

The proposed model is useful as a simpler and quicker way to obtain a pressure coefficient distribution for the cases where CFD tools are too labor intensive and the h/R ratio exceeds 10%.

Aeronautical applications that may benefit from the use of this model are especially the Upper Surface Blown aircraft but the theory may be also used for the predesign calculation of fluidic devices such as the ones described in (Olivotto, 2010).

BIBLIOGRAPHY

1. André Bakker, Applied Computational Fluid Dynamics Lecture 10, Turbulence Models, 2005.
2. Benner, S.I., , The Coandă Effect At Deflection Surfaces Widey Separated From The Jet Nozzle, University of Toronto.
3. CÎRCIU Ionică, Mircea BOŞCOIANU, An analysis of the efficiency of Coanda - NOTAR anti-torque systems for small helicopters, INCAS BULLETIN, Volume 2, Number 4/ 2010, pp. 81 – 88
4. Drăgan[a], V. A study of conventional upper surface blown wing configurations, Review of the Airfoirce Academy, Vol.X. no1, 2012
5. Drăgan[b], V. "Experimental Equations for Lift and Drag Coefficients of Entrainment Airfoils", International Review of Mechanical Engineering - September 2012, Vol. 6 N. 6
6. Drăgan[c], V. A New Mathematical Model for Coandã Effect Velocity Approximation, INCAS Bulletin, Vol.4 Iss.4
7. Frunzulică F., Dumitrache A., Preotu O.,Dumitrescu H. Control of two-dimensional turbulent wall jet on a coanda surface, PAMM Proc. Appl. Math. Mech. 11, 651 – 652 (2011).
8. Guo,B.D.;Liu,P.Q.;Qu,Q.L., Blowing Circulation Control on a Seaplane Airfoil, Recent progress in fluid dynamics research: Proceeding of the Sixth International Conference on Fluid Mechanics. AIP Conference Proceedings, Volume 1376, pp. 228-231 (2011).
9. Levinsky, S, T.T., Yeh, Analytical and Experimental Investigation Of Circulation Control By Means Of A Turbulent Coandă Jet.
10. Olivotto, Constantin , "Fluidic Elements based on Coanda Effect"INCAS BULLETIN, Volume 2, Number 4/ 2010, pp. 163 – 172
11. Roderick, W. E. B., Use of the Coanda Effect for the Deflection of Jet Sheets

over Smoothly Curved Surfaces, Part II, University of Toronto, Institute of Aerophys~cs, Technical Note No. 51, September 1961.

12. Saeed, B , G Gratton, Exploring the aerodynamic characteristics of a blown annular wing for vertical/short take-off and landing applications, Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, 2011
13. Slomski, Joseph and Tom Marino Navy successfully simulates effect that may improve low speed maneuverability, CFD Review 2003.
14. Wygnanski,Israel J., The Control of Separation from Curved Surfaces and Blunt Trailing Edges, AFOSR GRANT F 49620-00-1-0070, Final Technical Report, 2002
15. Yen.K.T.,, An Analysis Of The Flow Turning Characteristics Of Upper - Surface Blowing Devices For STOL Aircraft, 1982.