Thrust and jet directional control using the Coanda effect

Alexandru DUMITRACHE*[,1], Florin FRUNZULICA[2,1], Octavian PREOTU[4],
Tudor IONESCU[3,1]

*Corresponding author
[1]"Gheorghe Mihoc – Caius Iacob" Institute of Mathematical Statistics and
Applied Mathematics of the Romanian Academy,
Calea 13 Septembrie No.13, Sector 5, 050711, Bucharest, Romania
alex_dumitrache@yahoo.com*
[2]Department of Aerospace Sciences, "POLITEHNICA" University of Bucharest,
1-7 Gheorghe POLIZU St., sector 1, 011061, Bucharest, Romania
ffrunzi@yahoo.com
[3]Department of Automatic Control and Systems Engineering,
"POLITEHNICA" University Bucharest,
Splaiul Independentei 313, 060042, Bucharest, Romania
tudor.ionescu2006@upb.ro
[4]University of Craiova, Avionics Department,
Decebal 107 Bd., 200440, Craiova, Romania
opreotu@yahoo.com

DOI: 10.13111/2066-8201.2018.10.2.18

Received: 04 April 2018/ Accepted: 11 May 2018/ Published: June 2018

Abstract: *The application of the Coandă effect to the directional control of a jet or thrust is presented. Deviation of the thrust force by direct flow can be achieved by using the Coandă effect to change the angle of the primary jet engine exhaust nozzle. Major interest in the study of this phenomenon is caused by the possibility of using this effect for aircrafts with short take-off and landing, for thrust vectoring. The numerical investigations are performed using a RANS solver with an adequate turbulence model, showing a change of the jet direction. Thus, the conditions and the limits within which one can benefit from the advantages of Coandă-type flows are determined.*

Key Words: *Coandă effect, thrust vectoring, CFD.*

1. INTRODUCTION

Henri Coandă was the first man to notice this effect, in 1910, on the occasion of the first jet airplane flight, designed and piloted by himself, when the exhaust flames adhered to the airplane fuselage and burned it. In 1932, he obtained a patent in France for the invention "Procedure and device for the deflection of a fluid inside another fluid". This discovery is based on the phenomenon "deflection of a plane jet which enters another fluid in the vicinity of a convex wall", later known as the "Coandă effect". This effect allows a fluid jet to flow along a wall, which angles away from its previous flow direction (Fig. 1) [1, 2]. This concept and the corresponding advantages of its use, both in aerospace propulsion system and in industrial applications, have been studied for many years. Major interest in the study of this phenomenon is caused by the possibility of using this effect for aircrafts with short take-off and landing, for thrust vectoring.

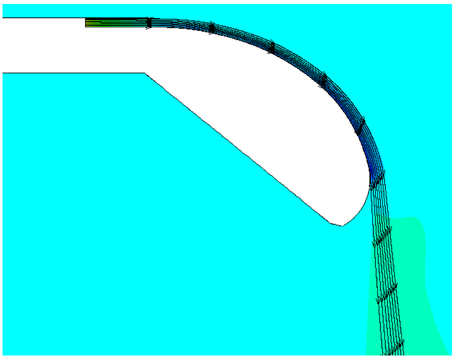

Fig.1 – The Coandă effect

After leaving the channel, the jet entrains through the friction effects the particles of the environment, in the zone where there is no flap and the fluid particles between the jet and the flap. If the flap is long enough, the place of the entrained particles in the domain between the flap and the jet can no longer be taken by the outer particles and hence the low pressure creates a flow deflection in the direction of the flap. In other words, if the surface is curvilinear convex, then a combination of skin friction, viscosity, and 'void effect' (near surface) results in a centripetal acceleration of the affected fluid, coexisting with a reduction in pressure because of inertia effects.

In the paper we present some applications of the Coandă effect in engineering, such as the jet or thrust vectoring. The diverse thrust vector systems can be classified according to the concept of control. The two main systems are the ones based on mechanical and on fluid control concepts. A non-moving thrust vector aircraft system has the advantage of the weight reduction and of easiest maintenance problems.

Investigations are made using Computational Fluid Dynamics, and the results highlight the positive and negative aspects of the technical solutions analyzed herein.

2. MATHEMATICAL MODEL

A. Dimensionless Form of Fluid Transport Equation

Dimensionless quantities are universal, and independent of operating variables, such as fluid, geometric scale, operating pressure, etc. Therefore, all parameters in the research are converted to the dimensionless form.

The fluid transport equations such as the mass (continuity), momentum, and energy conservation equations are presented in this section.

The mass conservation equation, or continuity equation, for the compressible flow is

$$\partial\rho/\partial t + \overline{\nabla}\cdot\left(\rho\overline{v}\right) = 0 \tag{1}$$

where: ρ = fluid density (kg/m^3), t = time (s), v = fluid velocity in a vector notation (m/s), $\overline{\nabla}$ = gradient operator.

The characteristic density and velocity are introduced to bring this equation into the dimensionless form. We define: ρ_c = characteristic (an inlet) density of the fluid (kg/m^3), U

= characteristic (an inlet) velocity of the fluid (m/s), t_c = characteristic time (s), and L = characteristic length = an inlet diameter of nozzle (m).

Then each term is converted to dimensionless form by multiplying and dividing each term by their characteristic parameters, and then rearranging the equation. Hence, the dimensionless form of this equation is

$$\partial\tilde{\rho}/\partial\tilde{t} + \tilde{\nabla}\cdot\left(\tilde{\rho}\tilde{v}\right) = 0 \tag{2}$$

The momentum conservation for compressible flows in dimensional form is [6]:

$$\begin{aligned} &\rho\frac{\partial}{\partial t}\left(\rho\bar{v}\right) + \bar{\nabla}\cdot\left(\rho\bar{v}\bar{v}\right) = -\bar{\nabla}P_{dyn} + \mu\nabla^2\bar{v} + 2\bar{\nabla}\mu\cdot\bar{\nabla}\bar{v} \\ &+\bar{\nabla}\mu\times\left(\bar{\nabla}\times\bar{v}\right) + \frac{1}{3}\mu\bar{\nabla}\left(\bar{\nabla}\cdot\bar{v}\right) - \frac{2}{3}\left(\bar{\nabla}\cdot\bar{v}\right)\bar{\nabla}\mu \end{aligned} \tag{3}$$

where: D/Dt = material derivative, P_{dyn} = dynamic pressure (Pa), μ = fluid viscosity ($N\cdot m$), ∇^2 = Laplacian operator.

Since the geometrical configuration of the ejector is axisymmetric, the continuity equation will be used in axisymmetric coordinates

$$\frac{\partial\rho}{\partial t} + \frac{\partial}{\partial x}\left(\rho\bar{v}_x\right) + \frac{\partial}{\partial r}\left(\rho\bar{v}_r\right) + \frac{\rho\bar{v}_r}{r} = 0 \tag{4}$$

and the mass conservation equation in the axisymmetric case is

$$\begin{aligned} &\frac{\partial}{\partial t}\left(\rho\bar{v}_x\right) + \frac{1}{r}\frac{\partial}{\partial x}\left(r\rho\bar{v}_x\bar{v}_x\right) + \frac{1}{r}\frac{\partial}{\partial r}\left(r\rho\bar{v}_t\bar{v}_x\right) = -\frac{\partial P}{\partial x} \\ &+\frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(2\frac{\partial\bar{v}_x}{\partial x} - \frac{2}{3}\left(\bar{\nabla}\cdot\bar{v}\right)\right)\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(\frac{\partial\bar{v}_x}{\partial r} + \frac{\partial\bar{v}_r}{\partial x}\right)\right] \end{aligned} \tag{5}$$

B. The Energy Equation

In compressible fluids, the energy equation is used in combination with the transport equations to calculate fluid properties.

The governing energy equation is presented (Fluent, 2001):

$$\frac{\partial}{\partial t}\left(\rho E\right) + \bar{\nabla}\cdot\left[\bar{v}\left(\rho E + P\right)\right] = \bar{\nabla}\cdot\left[k_{eff}\bar{\nabla}T - \sum_j h_j\bar{J}_j + \left(\bar{\bar{\tau}}_{eff}\bar{v}\right)\right] + S_h \tag{6}$$

where: E = internal energy (J), k_{eff} = effective conductivity (J/K), $\nabla T =$ total temperature difference (K), h_j = sensible enthalpy of species j (J), $\bar{J}_j =$ diffusion flux of species j, $\bar{\bar{\tau}}_{eff}$ = effective viscous dissipation ((Js)/m), and S_h = volumetric heat sources (J).

The equations can be spatially averaged to decrease computational cost, yet the averaging process yields a system with more unknowns than equations.

Hence, the unclosed system requires a model (e.g., turbulence, or subgrid scale) to make the problem well posed.

Such models are used in RANS and LES approaches to CFD. All the equations stated above are used to calculate fluid properties in CFD code, Fluent.

C. The Turbulence Modeling: SST Model Formulation

The basic idea behind the SST model is to retain the robust and accurate formulation of the Wilcox $k-\omega$ model in the near wall region, and to take advantage of the free stream independence of the $k-\varepsilon$ model in the outer part of the boundary layer.

To achieve this, the $k-\varepsilon$ model is transformed into a $k-\omega$ formulation by means of a function that has the value one in the near wall region and zero away from the surface. The final form is

$$\frac{\partial}{\partial x_j}\left(u_j k\right)=\frac{\partial}{\partial x_j}\left[\left(\nu+\sigma_{k1}\nu_t\right)\frac{\partial k}{\partial x_j}\right]+P_k-Y_k$$

$$\frac{\partial}{\partial x_j}\left(u_j \omega\right)=\frac{\partial}{\partial x_j}\left[\left(\nu+\sigma_{\omega}\nu_t\right)\frac{\partial \omega}{\partial x_j}\right]+\gamma P_\omega-Y_\omega+ \tag{7}$$

$$2\left(1-F_1\right)\sigma_{\omega 1}\frac{\nu_t}{k}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}$$

Model constants and implementation are presented in detail in [2], [4].

3. THRUST AND JET VECTORING

Mechanical deflection of the thrust involves engine nozzle deflection and, thus, physically modify of the main flow direction [3], [6,7]. In this case, complicatelly devices and actuators have to be used [8]. Fluid deflection of the thrust involves fluid injecting or removal from the boundary layer of a main jet to allow deflection. Alternative methods to mechanical deflection can be developed using a secondary jet to the entrainment of the main jet (Fig. 2). A deflection system of thrust fluid has the advantage of being easy, simple, inexpensive and without moving parts (fixed geometry), and can be implemented, taking advantage of a minimum radar detection of aircraft.

The component force of the thrust vector $T_0, F_{z,tv}$ produces a pitch moment, $M_{z,tv}$ around airplane's center of gravity, allowing the airplane to be controlled in flight. This force is adimensionalisated to obtain the thrust coefficient $C_z = F_{z,tv}/T_o$, where C_z depends on the angle of the thrust vector according to the equation $\delta_{tv} = \tan^{-1} C_z$. One must notice that the deviation of the thrust force implies a change of the drag force. When $d_{tv} > 0$, the real force of the primar jet will be smaller then the resultant thrust force $F_x < T_o$.

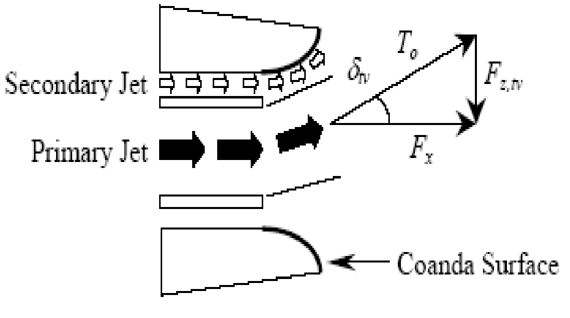

Fig. 2 – The concept of thrust deviation [9]

4. NUMERICAL MODEL AND RESULTS

The effect of the secondary flow on the main jet in the presence of a convex surface can be emphasized by numerical flow simulation on the configuration shown in Fig. 3. An axisymmetric Coandă ejector model is created with a structured and unstructured grid system with quadrilateral cells. The grid size is optimized to be small enough to ensure that the CFD flow results are virtually independent of the size, but large enough to ensure the model runs efficiently at an acceptable speed [5].

A non-uniform grid is selected since it provides the greatest control of the number of cells and their localized density. For optimal meshing, the grid density increased near the wall and in areas where flow gradients are steep. This is accomplished by applying weighting factors to increase the grid density at these areas.

The used mesh is divided in structured grid near the wall and unstructured grid otherwise. The y^+ values of the wall-next grid points are between 0.2 and 1, and the Δx^+ values are between 50 and 300.

The computational domain includes the adjacent regions of the configuration with the physical opening boundaries condition.

The RANS model with the second order spatial approximation model for flow equations, completed with the k-ω SST turbulence model, is used. The working fluid is the air, and the flow regime is assumed to be incompressible. The main jet has a speed of 25 m/s, with a moderate turbulence degree. By varying the secondary jet velocity (considered only the upper jet in the presented geometric configuration), the results shown in Fig. 4 are obtained.

a. b.

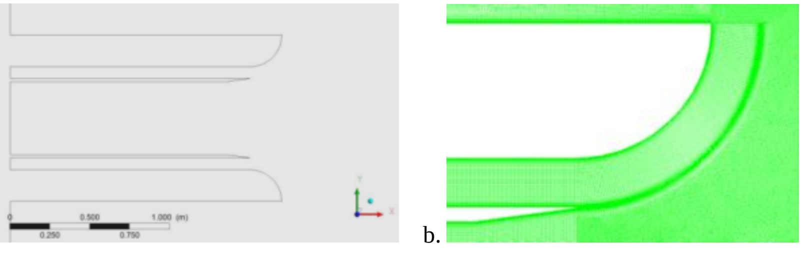

Fig. 3 – The model used to highlight the change of the jet direction: a. the model geometry; b. detail of the computational grid

Figs. 4 a1-b1 correspond to a speed V_s = 0 m/s for the secondary jet, a2-b2 for V_s = 30 m/s, and a3-b3 for V_s = 70 m/s. From Fig. 4 note that with the increase of the secondary jet velocity the deflection of the jet increases and the shape of the velocity profile in the main jet changes from a "full" velocity profile specific to the free jet to an asymmetric velocity profile with the maximum moved towards "the secondary jet". The deflection of the jet is "sustained" also by the shape of the exhaust surface (quarter circle).

Fig. 5 shows the variation of the main jet angle deflection in relation to the secondary jet velocity. If the secondary jet is accelerated over a Coandă surface, it will produce the local pressure drop and the appearance of the pressure gradient "perpendicular" to the main jet axis. This effect correlated with the friction effects leads to an increase in the fluid flow from the main jet to the secondary jet producing the change of the jet direction. Increasing the secondary jet velocity and the radius of curvature of the Coandă surface lead to a greater deflection of the main jet and obtaining the effect of the thrust vectorization. Note that for a secondary jet speed of over 60 m/s, the entrainment effect of the secondary jet decreases, the main jet being deflected insignificantly or not at all.

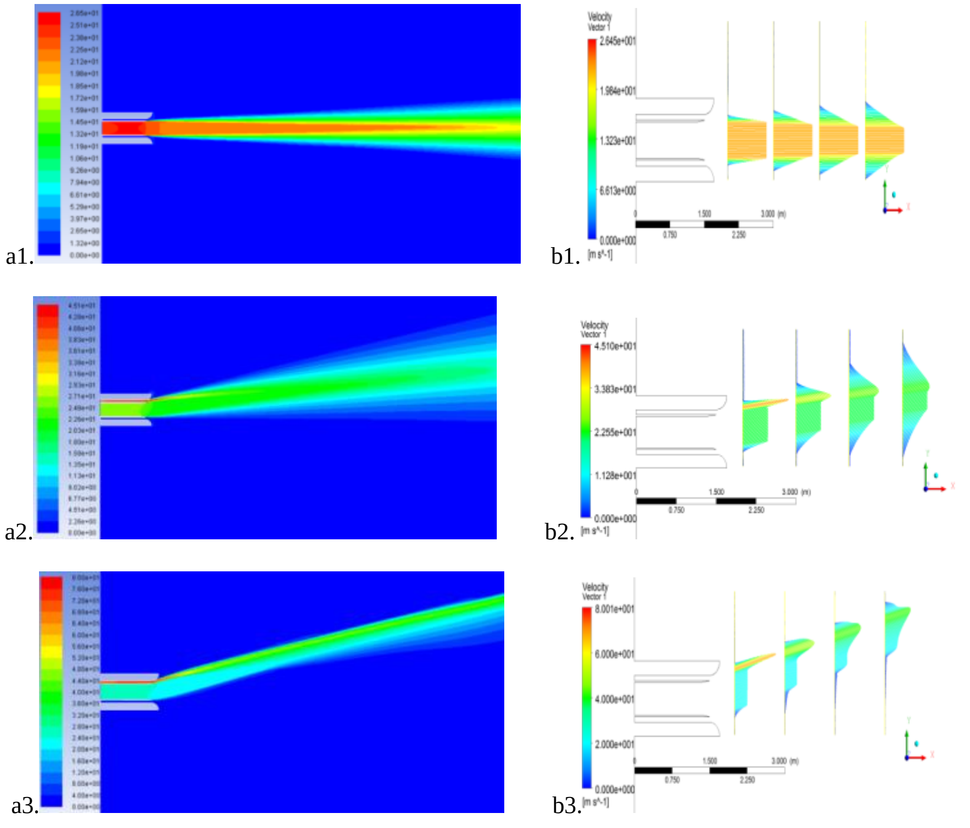

Fig. 4 – The effect of secondary jet on the primary jet (Vj = 25 m/s), the flow velocity field (a1-a3) and the velocity profiles (b1-b3) for 4 positions (2 m, 3 m, 4 m si 5 m)

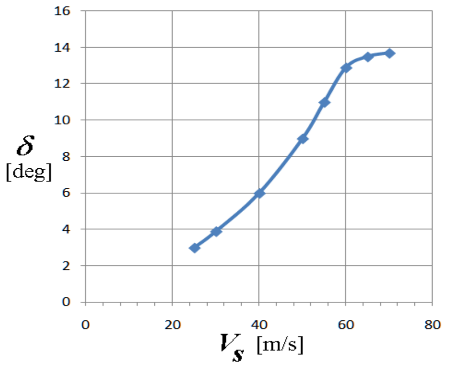

Fig. 5 – The deflection of the main jet as function of the velocity of the secondary jet

5. CONCLUSIONS

In this paper we have presented an application of the Coandă effect in civil engineering/aeronautics regarding the thrust or jet vectoring. Deviation of the thrust force by direct flow can be achieved by using the Coandă effect to change the angle of the primary jet engine exhaust nozzle. This effect correlated with the friction effects leads to an increase in the fluid flow from the main jet to the secondary jet producing the change of the jet direction.

We have numerically analyzed the effect of secondary jet on the primary jet and the deflection of the main jet as function of the velocity of the secondary jet has been obtained. Thus, the conditions in which the Coandă effect can successfully be used to produce a controlled deflection of a traction have been determined. The effect of a secondary jet with a speed of over 60 m/s has little effect on the main jet deflection. To achieve a large deflection angle for the jet, the side effects of the jet development must be controlled.

Further investigation can be done using synthetic jets to obtain an augmenting vectoring effect.

To validate these results, experimental investigations are required on a model, for comparison.

The Coandă effect offers extensive perspectives in aeronautics and non-aeronautics applications.

ACNOWLEDGEMENT

This work has been supported by the grant of the Romanian National Authority for Research and Innovation, CNCS/CCDI-UEFISCDI, project number PN-III-P2-2.1-PED-2016-0376, within PNCDI III.

This article is an improved version of the science communication with the same title, presented in *The 37th edition of the Conference "Caius Iacob" on Fluid Mechanics and its Technical Applications*, 16-17 November 2017, Bucharest, Romania, (held at INCAS, B-dul Iuliu Maniu 220, sector 6), *Section 1. Basic Methods in Fluid Mechanics*.

REFERENCES

[1]. V. A. Drăgan, New Mathematical Model for Coandã Effect Velocity Approximation, *INCAS Bulletin*, (online) ISSN 2247–4528, (print) ISSN 2066–8201, ISSN–L 2066–8201, vol. **4**, Issue 4, pp. 85 – 92, DOI: 10.13111/2066-8201.2012.4.4.7, 2012.

[2] A. Dumitrache, F. Frunzulica and T. Ionescu, *Mathematical Modelling and Numerical Investigations on the Coanda Effect*, chapters (5) in book: *Nonlinearity, Bifurcation and Chaos - Theory and Applications*, book edited by Jan Awrejcewicz and Peter Hagedorn, InTech, Published: October 24, 2012, pp. 101-132, 2012.

[3] J. C. Páscoa, A. Dumas, M. Trancossi, P. Stewart and D. Vucinic, A review of thrust-vectoring in support of a V/STOL non-moving mechanical propulsion system, *Central European Journal of Engineering*, vol. **3**, no. 3 pp. 374-388, 2013.

[4] F. R. Menter, Eddy viscosity transport equations and their relation to the k-ω model, *ASME Journal of Fluids Engineering*, vol. **119**, pp. 876- 884, 1997.

[5] * * * Ansys Fluent 13.0. User's Guide.

[6] G. S. Jones, Pneumatic Flap Performance for a Two-Dimensional Circulation Control Airfoil, in *Applications of Circulation Control Technologies – Progress in Astronautics and Aeronautics*, vol **214**, American Institude of Aeronautics and Astronautics, 2006.

[7] W. J. Baker and E. G. Paterson, Simulation of Steady Circulation Control for the General Aviation Circulation (GACC) Wing, in *Applications of Circulation Control Technologies – Progress in Astronautics and Aeronautics*, vol **214**, American Institude of Aeronautics and Astronautics, 2006.

[8] R. J. Englar, Overview of Circulation Control Pneumatic Aerodynamics: Blown Force and Moment Augmentation and Modification as Applied Primarly to Fixed-Wing Aircraft, in *Applications of Circulation Control Technologies – Progress in Astronautics and Aeronautics.* vol **214**, American Institude of Aeronautics and Astronautics, 2006.

[9] M. S. Mason and W. J. Crowther, *Fluidic Thrust Vectoring of Low Observable Aircraft*, CEAS Aerospace Aerodynamic Research Conference, June 2002.