# The influence of convective exchanges on coandã effect

TRANCOSSI, Michele and PASCOA, J

# The influence of convective exchanges on Coandã effect

Michele TRANCOSSI*,[1], Jose PASCOA[2]

*Corresponding author
*,[1]Henri Coanda Labs LLC, Italian Headquarter,
Via Milazzo, 8, Parma 43125, Italy,
mtrancossi@gmail.com
[2]C-MAST – Center for Mechanical and Aerospace Science and Technologies,
Universidade da Beira Interior,
Convento de Sto. António, 6201-001 Covilhã, Portugal,
pascoa@ubi.pt

DOI: 10.13111/2066-8201.2019.11.4.17

*Received: 07 July 2019/ Accepted: 23 October 2019/ Published: December 2019*


***Abstract:*** *Modeling Coandã effect has been a fundamental issue in fluid dynamic research in the XX century. It has lost some interest because of the improvement in CFD, even if it could be still important in the area of the preliminary design of aerodynamic devices that benefits of fluid deflection by convex surfaces. An effective model of Coandã effect has not been defined, and fundamental questions are still open. The influence of convective heat exchange on Coandã adhesion of a fluid stream on a convex surface in the presence of a temperature gradient between the fluid and the convex surface is a problem, which affects many practical cases, but it is still marginally approached by scientific literature. This paper aims to start an effective research direction on the effects of convective heat exchange on Coandã effect. It approaches the problem with a set of CFD simulations. It analyses the previous hypotheses, which are based on Prandtl number and evidences the need of a more effective model that accounts also for the Reynolds number.*

***Key Words:*** *fluid dynamics, aerodynamics, Bejan number, fundamental equations, the first law of thermodynamics, the second law of thermodynamics, constructal law*

## 1. INTRODUCTION

This paper is a step toward an effective analysis of the Coandã effect in the presence of heat exchanges between the fluid stream and the convex surfaces. It considers the previous results which have been obtained during the EU FP7 ACHEON (Aerial Coandã High Efficiency Orienting-jet Nozzle) Project [1-4]. It analyses the influence of thermal interaction with the wall and the consequent convective heat transfer on Coanda effect adhesion on a high speed fluid flow.

It can be remarked that Coandã effect is almost ubiquitous into fluid dynamics and aeronautics. It applies to fluidic devices and turbo-machines such as Guo et al. [5] and Dragan [6, 7] demonstrate. Most applications rely on peculiar fluidic jets blowing tangentially to a curved surface to create lift directly or indirectly, deflecting the direction of a jet stream and reducing vorticity.

The development of fluid dynamics and the increased dependency on CFD have reduced the necessity of theoretical models in recent years, notwithstanding their importance as a frame for which it would be useful as a tool for a better preliminary design and reduction of the overall design costs and times.

Most of the studies on the development of the Coandã effect consider isothermal fluid dynamic phenomena but neglect the effects of the thermal interactions that may occur between the jet stream and the Coandã surface. This paper focuses on determining how these effects can be accounted for a better and more complete explanation of the Coandã effect and the complex equilibrium conditions.

The above considerations force to consider different modelling effort, which has been produced and verify how they can be integrated with essential elements of heat exchange.

Newman [8] has experimentally investigated a two-dimensional, incompressible, turbulent jet flowing around a circular cylinder. Newman’s setup is presented in Figure 1 because of its importance in the analysis of the Coandã effect. He obtained the following expression of the angle of separation the impinging jet from the Coandã surface:

$$\theta_{sep} = f\left[\frac{(p_0 - p_\infty)b \cdot a}{\rho U^2}\right] \tag{1}$$

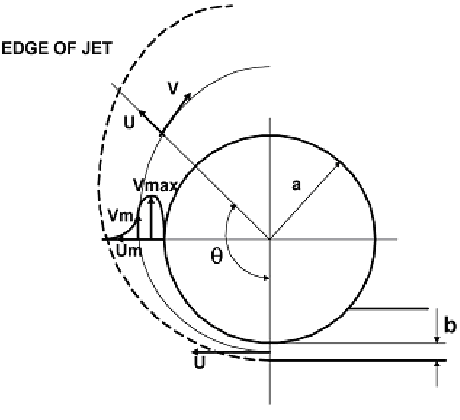



Fig. 1 – Newman’s experimental setup

Bradshaw [9] explained the Coandã Effect in terms of inviscid irrotational flow. He has assumed that the flow is initially inviscid and the relation between pressures can be obtained by Bernoulli equation:

$$p_0 = p_\infty - \frac{\rho \cdot U^2 \cdot b}{a} \tag{2}$$

In particular, the detachment condition for the jet is given by:

$$\frac{\rho \cdot U^2 \cdot b}{a} \le p_\infty \tag{3}$$

Dragan [6] has demonstrated that the model by Banner [10] works well to describe Coandã effect under certain conditions. Banner has considered the balance between the pressure and centrifugal forces exerted on an infinitesimal control volume dm in the Coandã stream. He estimated the average pressure coefficient across a circular ramp.

If *b/a* ratio is much smaller than one, the balance between the pressure forces and the centrifugal forces acting upon the volume can be expressed by equalizing pressure and centrifugal forces.

$$F_c = \frac{\rho R \cdot d\theta \cdot da \cdot u^2}{R} = F_p = Ldp = a \cdot d\theta \cdot dp \quad (4)$$

It is then possible to determine equation (5), which expresses the pressure drop of the jet:

$$\Delta p = p_{static,\,jet} - p_\infty = \frac{2b}{a} \cdot \frac{\rho U^2}{2} \quad (5)$$

Consequently, it is possible to determine the thrust of an element of the jet:

$$F_h = h\rho U^2 \quad (6)$$

Hence, it is possible to determine the pressure coefficient

$$C_p = \frac{\Delta p}{(\rho v^2)/2} = -\frac{2 \cdot b}{a} \quad (7)$$

Roderick [11] obtained a similar result some years after for thicker jets (where the thickness *b* of the impinging jet is comparable with the curvature radius R).

$$C_p = -\frac{2 \cdot b}{a}\left(1 + \frac{b}{2 \cdot a}\right) \quad (8)$$

More recently, some modelling efforts have been produced during the ACHEON Project by Trancossi et al. [12, 13], who has approached an analytical model based on Bernoulli equation coupled with polynomial boundary layer models. Das at al. [14] and Subhash et al. [15] has proposed CFD based models.

It must be remarked that the scientific literature analyses the pure fluid dynamic behaviour of the Coandã effect and neglects the thermal effects that may happen between fluid and convex surfaces.

During the ACHEON project, a preliminary attempt of describing the relation of fluid flow adhesion and surface temperature on a convex surface has been produced by Dumas et al. [16].

They have preliminary analyzed the effects of the temperature gradient between the convex surface and the Coandã flow.

They have analyzed how increasing the temperature of the convex surface influences the flow behaviour and have identified two possible mechanisms, which could govern the fluid flow in the presence of a temperature gradient.

One is based on variable Prandtl number, and the second is based on constant Prandtl number (thermal diffusivity) effect.

In particular, they have verified that the increment of the thermal diffusivity facilitates the jet stream adhesion and elongates the contact length before the separation of the boundary layer, while other mechanism triggers the earlier detachment of the flow from the curved surface.

Dumas et al. have attempted to use CFD analysis to understand the effect of heat exchanges on the stability of Coandã effect governed fluid dynamics adhesion. Even still limited to a single temperature gradient case, this preliminary analysis has the merit of evidencing the disturbances, which are generated by heating the surface and producing a temperature gradient.

This analysis has, in any case, the merit of opening a new field of research with interesting perspectives through better comprehension of the Coandã effect.

## 2. FUNDAMENTALS ON CONVECTIVE HEAT EXCHANGE

The rate of convective heat transfer between the fluid and the wall is given by:

$$\dot{q}_{conv} = h\left(T_w - T_\infty\right) \tag{9}$$

If it is assumed the no-slip condition, the heat transfer between a solid surface and an adjacent fluid layer is by pure conduction, since the fluid is motionless. Thus, it results:

$$\dot{q}_{conv} = h(T_w - T_\infty) = \dot{q}_{cond} = k_{fluid} \left.\frac{\partial T}{\partial y}\right|_{y=0} \tag{10}$$

The Boundary layer region is the region where the viscous effects and the velocity changes are significant, the inviscid region is the region in which the frictional effects are negligible, and the velocity remains almost constant. The friction between two adjacent layers is shear stress:

$$\tau_s = \mu \left.\frac{\partial V}{\partial y}\right|_{y=0} \tag{11}$$

Viscosity is a measure of fluid resistance to flow and is a function of temperature. The surface shear stress can also be determined from:

$$\tau_s = C_f \frac{\rho U_\infty^2}{2} \tag{12}$$

The flow in the boundary layer starts as smooth and streamlined laminar flow. At some distance from the leading edge, the flow turns turbulent, and it is characterized by highly disordered motion.

The transition from laminar to turbulent flow occurs over some region, which is defined as the transition region. The velocity profile in the laminar region is approximately parabolic and becomes flattered in a turbulent flow.

This paper aims to move toward an extension of the analysis, as mentioned above by Dumas et al. with a more effective analysis of the boundary layer phenomena. This effort requires considering some further fundamentals on fluid.

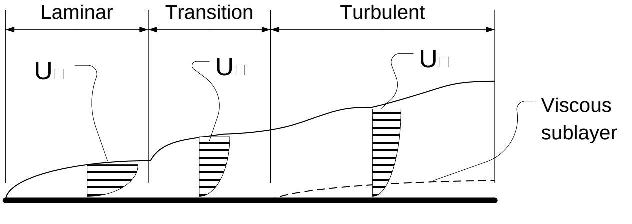



Fig. 2 – Velocity boundary layer

Two dimensionless magnitudes can describe convective heat exchange:

1. *Nusselt number*: non-dimensional heat transfer coefficient

$$Nu = \frac{\dot{q}_{conv}}{\dot{q}_{cond}} = \frac{h \cdot L}{k} \tag{13}$$

where $L$ is the length of the solid boundary for a flat plate.

The Nusselt number represents the enhancement of heat transfer through a fluid because of convection relative to conduction across the same fluid layer.

2. *Reynolds number*: the ratio of inertia forces to viscous forces in the fluid

$$\mathrm{Re_L} = \frac{\rho UL}{\mu} = \frac{UL}{\nu} \tag{14}$$

At large Re numbers, the inertia forces, which are proportional to the density and the velocity of the fluid, are large relative to the viscous forces. The viscous forces cannot prevent the random and rapid fluctuations of the fluid (turbulent regime).

1. *Prandtl number*: is a measure of the relative thickness of the velocity and thermal boundary layer.

$$\mathrm{Pr} = \frac{\nu}{\alpha} = \frac{\mu / \rho}{k / (\mathrm{c_p}\, \rho)} = \frac{c_p \mu}{k} \tag{15}$$

It is given by the ratio of molecular diffusivity of momentum and thermal diffusivity. Prandtl number directly relates to the thickness of the thermal boundary layer. A thermal boundary layer develops when fluid at a specific temperature flows over a surface, which is at different temperature. The thickness of the thermal boundary layer $\delta_t$ is defined as the distance at which:

$$\frac{T - T_w}{T_\infty - T_s} = 0.99 \tag{16}$$

Prandtl number describes the relative thickness of the velocity and the thermal boundary layers. For high Prandtl number fluids, i.e. oils, heat diffuses much slower than the momentum and the thermal boundary layer is contained within the velocity boundary layer.

The different values of Prandtl number for air allow determining the expected evolution of relative thickness of the velocity and thermal boundary layer. Values have been plotted in Fig..

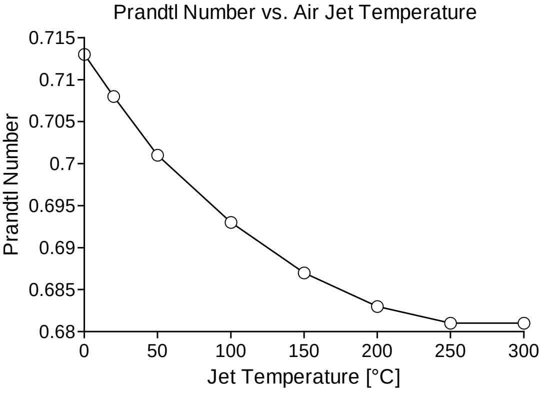



Fig. 3 – Prandtl number of air

In the case of Turbulent Flow, the local friction coefficient and the Nusselt number at any location *x* for turbulent flow over a flat isothermal plate are:

$$Nu_x = \frac{hx}{k} = 0.0296\, \mathrm{Re}_x^{4/5}\, \mathrm{Pr}^{1/3} \tag{17}$$

which is valid for $0.6 \leq Pr \leq 60$ and $10^5 \leq Re \leq 10^7$, and

$$C_{f,x} = \frac{0.0592}{\mathrm{Re}_x^{1/5}} \tag{18}$$

which is valid for $10^5 \leq Re \leq 10^7$. The average friction coefficient and Nusselt number over the isothermal plate in the turbulent region are:

$$Nu_L = \frac{hx}{k} = 0.037\,\mathrm{Re}_L^{4/5}\,\mathrm{Pr}^{1/3}, \tag{19}$$

and

$$C_{f,L} = \frac{0.074}{\mathrm{Re}_L^{1/5}} \tag{20}$$

More detailed studies force to consider that air is compressible fluid and the temperature influences its speed, because of density variation, which is caused by heating and influences the Reynolds number.

## 3. METHODS

An extensive CFD campaign has been performed assuming a geometry which is derived from the one adopted by Dumas et al. (Fig. 4a.) and their geometric setup, which is presented in Fig. 4:

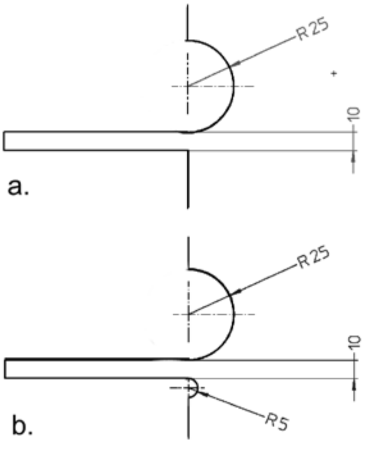



Fig. 4 – Comparison between Dumas at al. and actual geometry

The geometry has been implemented (fig. 4b.) by the introduction of a small radius at the lower edge of the jet stream outflow. This modification has been implemented because of the choice of adopting a new meshing schema. It is based on an unstructured quadrilateral mesh with a refinement near the walls [17]. This choice allows a better quality mesh by using the desired quadrilateral unstructured schema. Besides, the model has been improved by the adoption of a more effective hemi-circular outlet that allows a significant reduction in terms of the number of cells, without affecting the problem solution.

In particular, the quadrilateral mesh schema is encouraged by the adoption of a different CFD code. In particular, the new analysis has been produced by EasyCFD [18] a CFD software tool that allows numerical simulations of 2D and 3D axis symmetric fluid flow in a boundary-fitted mesh [19, 20]. The domain has been presented in Fig. a., which shows the nature of the boundaries.

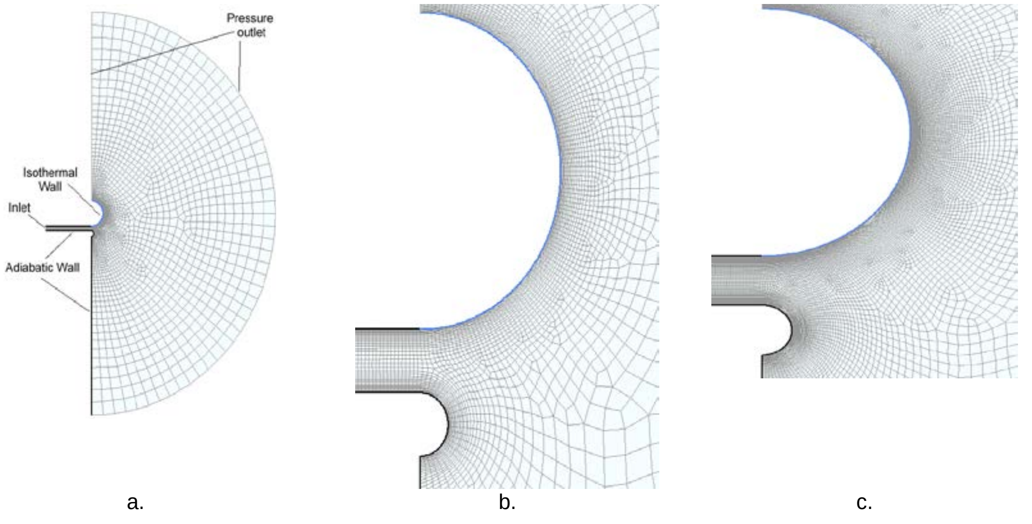



Fig. 5 – a. Schema of the domain; b. Detail of mesh with (coarse mesh); c. Detail of mesh at high resolution.

The Coandã surface has a radius of curvature of 0.025 m, and the thickness of the jet slot is 0.01 m. The preliminary independent grid check has been performed according to ERCOFTAC [21, 22] guidelines as applied by Rizzi and Vos [23] and Celik et al. [24].

The following conditions have been defined:

1. Physical conditions
   - Reference pressure: 101,325 Pa
   - fluid: air
   - density: 1,1884kg/m$^3$;
   - dynamic viscosity: 1,824e-5Ns/m$^2$;
   - thermal conductivity: 0,0257355 W/(mK);
   - expansion coefficient: 3,4113e-3 1/K;
   - Cp: 1006 J/(kg K);
   - Pr: 0.713;
2. Turbulence model: k-ε [25-26];
3. Convergence criteria:
   - Residual M: $1x10^{-7}$;
   - Residual U: $1x10^{-7}$;
   - Residual W: $1x10^{-7}$;
   - Residual K: $1x10^{-6}$;
   - Residual D: $1x10^{-6}$;
   - Residual T: $1x10^{-6}$.

Second order upwind scheme has been used for the discrimination of both the momentum equation and the k-ε turbulence model. Numerical stability has been approached according to well-accepted standards. A numerically stable grid has been determined through the numerical computation of the grid at different refinement level of the grid at the curved surface. The primary reference for this analysis has been the Policy Statement by the ASME Journal of Fluid Mechanics [27].

The problem involves Coandã adhesion and fluid attraction. It requires a more in-depth investigation by considering necessary supplementary sources [28, 29]. The flow field has been computed on different grids, each obtained from the previous by reducing the grid size (in *x* and *y* directions) by a factor of 0.75. No analysis in *z* coordinate direction is necessary being a 2D problem. Table 1 reports the basic grid information and the resulting pressure recovery computed from the solutions.

The following geometric parameters have been assumed:

1. Isothermal channel wall area: 0,0785377 $m^2$;
2. Inlet area: 0.01 $m^2$.

When the grid resolved the viscous sub-layer until y+ value lower than two, the jet deflection angle appears independent from the size of the grid. Pressure gradient term has been discretized by the second order upwind method. The unsteady term has been discretized using first-order implicit method taking advantage of unconditionally stable for time step size, which has been as evaluated $\Delta t = 1\times10^{-03}$ s [30]. Each solution was converged adequately with respect to the iterations. The column indicated by "spacing" is the spacing normalized by the spacing of the finest grid on the Coandã surface.

Table 1 shows the shear stress, which is associated with different grid dimensions. As the grid spacing reduces, the Shear Stress approaches an asymptotic zero-grid spacing value. It can be calculated the order of convergence that results in $L_{conv}$=0.5063%, which is much lower than the theoretical order of convergence is $L_{th,conv}$=2.0.

Table 1. Grid convergence analysis

| Grid | Norm. Grid Spacing | Wall temp. | Inlet air speed | Inlet Temp. | Wall shear stress |
|---|---|---|---|---|---|
| | (mm) | (°C) | (m/s) | (°C) | (N/$m^2$) |
| 1 | 1.3 | 20 | 10 | 20 | 0.2167647 |
| 2 | 1.95 | 20 | 10 | 20 | 0.2160171 |
| 3 | 2.6 | 20 | 10 | 20 | 0.2149289 |
| 4 | 3.9 | 20 | 10 | 20 | 0.2120638 |
| 5 | 5.2 | 20 | 10 | 20 | 0.2080674 |
| Wall Shear Stress convergence ratio $S_{conv}$ | | | | | 1.01031 |
| Theoretical Shear Stress $L_{rh}$ | | | | | 0.2181 |
| Grid Convergence Index GCI1,2 (FS=1.5) | | | | | 0.3461% |
| Grid Convergence Index GCI2,3 (FS=1.5) | | | | | 0.5063% |
| Grid Convergence Index GCI3,4 (FS=1.5) | | | | | 1.3511% |
| Grid Convergence Index GCI4,5 (FS=1.5) | | | | | 1.9207% |

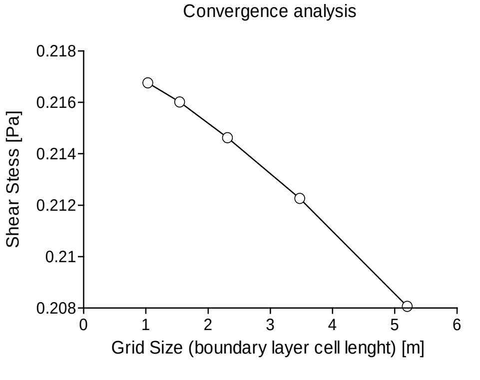



Fig. 6 – Wall shear stress values for convergence analysis

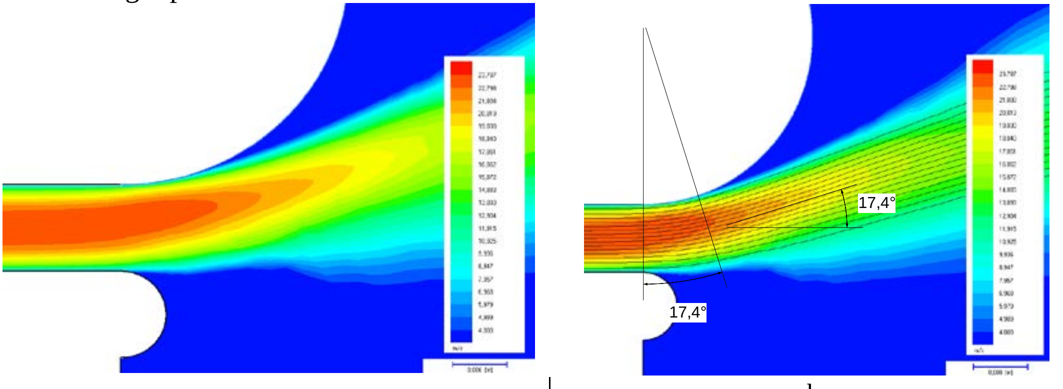
## 4. RESULTS

Dumas at al. have considered only the case of a cold fluid jet and a hot wall. They have assumed the temperature of the curved surface at 100°C and 600°C.

The present study extends the previous study by Dumas et al. by considering two different complementary cases:

1. the case of heating the Coandã surface from 20 to 800°C assuming a constant temperature inlet gas (T=20°C);
2. the case of keeping the wall at constant temperature and of heating the fluid inlet from 20 to 800°C.

The effect of the two different temperature gradients above-identified is analyzed and discussed. Fig. 7a shows the velocity contours in one of the analyzed cases. It allows explaining the adopted process for determining the angles of the jet stream. The simplest way of determining the angles is to assume the exact direction of the streamlines near the nozzle (Fig. 7b). They can be determined by using the streamlines and considering the velocity on the almost straight part of the streamlines as above.




a. b.

Fig. 7 – a. sample of velocity contours; b. sample of velocity contours and streamlines plotting and identification of angles

It must also be remarked that EasyCFD allows exporting the values of an arbitrary set of points on a line on a spreadsheet.

In this operating way, it has been possible to determine a more precise solution to the above geometry problem. The results have been reported in Table 1 and shown graphically in Fig. 11 and 12. In the case of a heated surface, such as in Dumas et al., the velocity contours show that the induced velocity at the curved surface seems influenced by the pressure gradient.

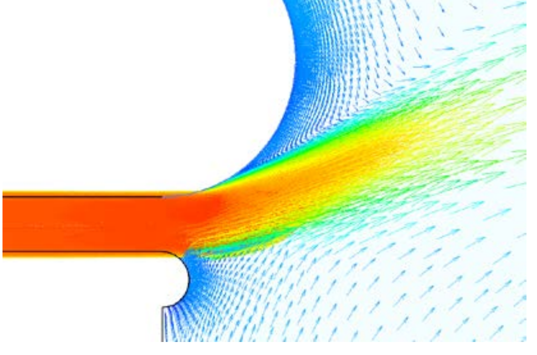


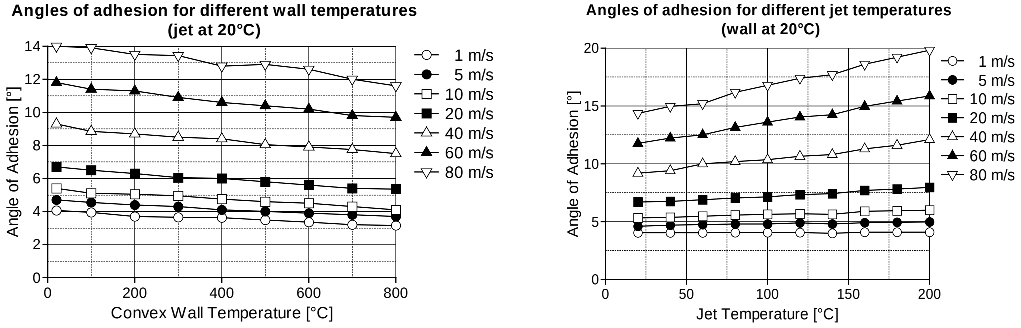
Fig. 8 – Velocity vectors showing main fluid streams and surrounding fluid suction effects

Fig. 8 shows the velocity vectors and allows verifying in detail the very low vorticity that is present around the main Coandã jet. Besides, it shows the presence of the suction effect, which is produced by the high-speed jet. The results confirm the research by Dumas et al. by evidencing a small vortex formation at the outer core of jet, which is evidenced by the irregular shape of velocity contours.

It is confirmed that the increments of the temperature of curved surface induced the earlier detachment of the jet. The results are presented in Fig. 9.

This paper presents a second case 2, the one of a heated fluid jet. In this case, the convex wall is at a uniform temperature, and the jet can assume different temperatures.

This case has some relevance in the field of industrial high-temperature plants, aeronautic propulsion and combustion. It must be remarked that it, has not been considered by Dumas et al.




a. b.

Fig. 9 – CFD results: a. heated Coandã wall and jet at 20 (°C); b. results for heated fluid and wall at 20 °C.

The velocity has been computed in terms of a function of mass flow inlet. The equivalence between the original velocity and the corresponding mass flow is indicated in the legend of Fig. 9 that reports the results in the case of a heated fluid. The solutions have been limited in terms of the jet speed at 80 m/s. The temperature of the jet has been limited to 200°C.

## 5. ANALYSIS OF THE RESULTS

The preliminary discussion in the introductive part of the paper shows that the analysis by Dumas et al. could need further improvements, being based on Prandtl number Pr, only. Pr has the physical meaning of the ratio between momentum diffusivity and thermal diffusivity. This choice is probably caused by focusing on temperature gradient and not on the complex set of phenomena that governs the Coandã effect. Hence, it must be observed that the thermal diffusivity has a much higher sensibility to variation in the temperature gradient rather than in thermal diffusivity. Hence, it can be observed that for the same fluid Pr can change with temperature. On the other side, it can be observed that the use of Prandtl number lacks in terms of generality because it is strongly dependent on the nature of the fluid. Besides, the critical magnitude that governs the fluid adhesion by Coanda effect is momentum, as shown by Fig. 5. An increase in the temperature gradient between the wall of the Coandã surface and the fluid jet appears facilitating the detachment of the jet. However, the temperature of the curved surface appears, enhancing the thermal diffusivity of the flow and increasing the momentum of the jet. In the opposite case, when the jet is at a higher temperature than the wall of the Coandã surface, the fluid adhesion to the surface seems favoured.

According to Dumas et al., it appears evident that two different mechanisms govern the Coandã adhesion. One mechanism enhances the initial adhesion to the curved surface and can inhibit the earlier detachment of the jet. On the other, it accelerates the late detachment of the jet. According to Dumas et al., this mechanism is based on phenomena that take place at variable Prandtl Number. A second mechanism can be envisaged and relates to constant Prandtl number. They have claimed that the mechanism that variable Prandtl number seems dominant if compared to phenomena that take place at variable Prandtl number. Hence, the results are in line with the general theory of convective heat transfer [31, 32], which affirms that the convective coefficient increases when the velocity increases. The preliminary analysis, which has been described in the introduction, encourages a slightly different model. It allows envisaging a more complex relationship that also involves the dynamic parameters of the jet. In particular, the adhesion angle seems, in the considered velocity range, almost proportional to both jet velocity and dependent on the difference of temperature between the jet and the Coandã surface. The temperature profile remains almost the same at all velocities. It allows thinking that a relation of proportionality directly relates the heat, which is subtracted from the jet. Equation (4) presents the equilibrium of an elemental volume of fluid in the jet stream.

It is consequently evident that the phenomena, which are described, are a function of Reynolds number. Consequently, they depend on the density and speed of the fluid, which are related by the equation of conservation of mass. A more accurate analysis needs to consider the different thicknesses of both velocity and thermal boundary layers during their relative development but also requires considering both Nusselt and Reynolds number.

## 6. CONCLUSIONS

Most of the studies on Coanda effect consider isothermal fluid dynamic phenomena but neglect the effects of the thermal interactions that may occur between the jet stream and the Coandã surface. Dumas et al. [16] have presented a numerical study of the Coandã adhesion of a fluid stream to a convex surface. They have considered the case in which the convex surface has a higher temperature with respect to the jet stream. Their analysis limits to a preliminary hypothesis according to which the behavior of the Coanda effect in the presence of this temperature difference is a function of Prandtl number. This paper extends the numerical analysis of a similar model to the case in which the jet has a higher temperature with respect to the convex surface. Notwithstanding the adoption of a different CFD software, the numerical results appear in line with the one Dumas et al., but a much larger number of cases is considered.

A broader spectrum of analysis evidences that the model based on Prandtl number (fixed and variable Prandtl number) only allows understanding the phenomena but does not seem exhaustive. In particular the definition of the variable Prandtl number seems problematic and is already included in the dynamic equilibrium of pressures and forces that govern Coandã effect. The performed simulations, are non-sufficient to advance exhaustive hypotheses toward an exhaustive modelling of Coandã effect in the presence of heat exchange phenomena, even if it produces some consideration that may help future scientific work in toward a better comprehension of these phenomena. In conclusion, this paper aims to be a starting point towards a more exhaustive comprehension of the phenomena that influence the Coandã adhesion in the presence of convective heat exchange phenomena.

## REFERENCES

[1] M. Trancossi, and A. Dumas, *ACHEON: Aerial Coanda High Efficiency Orienting-jet Nozzle*, SAE Technical Paper No. 2011-01-2737, 2011.

[2] M. Trancossi, *An overview of scientific and technical literature on Coanda effect applied to nozzles*, No. 2011-01-2591, SAE Technical Paper, 2011.

[3] J. C. Páscoa, G. Ilieva, A. Dumas, and M. Trancossi, A review of thrust-vectoring in support of a V/STOL non-moving mechanical propulsion system, *Central European Journal of Engineering* **3**.3:374-388, 2013.

[4] A. Dumas, J. Pascoa, M. Trancossi, A. Tacchini, G. Ilieva, and M. Madonia, Acheon project: A novel vectoring jet concept, In *ASME 2012 International Mechanical Engineering Congress and Exposition*, *American Society of Mechanical Engineers*, 499-508, 2012.

[5] B. D. Guo, P. Q. Liu, Q. L Qu, 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**:228-231, 2011.

[6] V. Dragan, A new mathematical model for high thickness Coanda effect wall jets, *Review of the Air Force Academy* 1: 23-28, 2013.

[7] V. Dragan, Reynolds number calculation and applications for curved wall jets, *INCAS Bulletin* **6**.3:35, 2014.

[8] B. G. Newman, *The Deflexion of Plane Jets by Adjacent Boundaries- Coanda Effect, Boundary Layer and Flow Contro*l, edited by G. V. Lachmann, Vol. **1**, Pergamon Press, Oxford (1961), pp. 232-264.

[9] P. Bradshaw, *Effects of Streamline Curvature on Turbulent Flow*, AGARDograph AG-169, 1960.

[10] S. D. Benner, *The Coanda Effect at Deflection Surfaces Widely Separated from the Jet Nozzle*, Toronto Univ. Downsview (Ontario) Inst for Aerospace Studies, no. UTIAS-TN-78, 1964.

[11] W. E. B. Roderick, *Use of the Coanda Effect for the Deflection of Jet Sheets over Smoothly Curved Surfaces*, Part II. Toronto Univ. Inst for Aerospace Studies, Downsview (Ontario), no. UTIAS-TN-51, 1961.

[12] M. Trancossi, A. Dumas, and D. Vucinic, *Mathematical modeling of Coanda effect*, SAE Technical Paper No. 2013-01-2195, 2013.

[13] M. Trancossi, A. Dumas, S. S. Das, and J. Pascoa, Design methods of Coanda effect nozzle with two streams, *Incas Bulletin* **6**.1: 83, 2014.

[14] S. Das, M. Abdollahzadeh, J. Pascoa, A. Dumas, and M. Trancossi, Numerical modeling of coanda effect in a novel propulsive system, *The International Journal of Multiphysics* **8**.2, 2016.

[15] M. Subhash, M. Trancossi, and J. Páscoa, *An Insight into the Coanda Flow Through Mathematical Modeling*, In Modeling and Simulation in Industrial Engineering, Springer, Cham, 101-114, 2018.

[16] A. Dumas, M. Subhash, M. Trancossi, and J. Pascoa, The influence of surface temperature on Coanda effect, *Energy Procedia* **45**:626-634, 2014.

[17] T. D. Blacker, and M. B. Stephenson, Paving: a New Approach to Automated Quadrilateral Mesh Generation, *International Journal for Numerical Methods in Fluids*, Vol. **32**: 811-847, 1991.

[18] A. M. G. Lopes, *EasyCFD ver. 4.4.1*, ADAI - Departament of Mechanical Engineering, University of Coimbra, Portugal. http://www.easycfd.net/ (retrived May 2019).

[19] A. M. G. Lopes, A versatile software tool for the numerical simulation of fluid flow and heat transfer in simple geometries, *Computer Applications in Engineering Education* **18**.1:14-27, 2010.

[20] A. M. G. Lopes, A Numerical Procedure for 2D Fluid Flow Simulation in Unstructured Meshes, *Numerical Simulation-From Brain Imaging to Turbulent Flows*, IntechOpen, 2016.

[21] M. Casey, and T. Wintergerste, and I. Sulzer, *ERCOFTAC special interest group on quality and trust in industrial CFD*, Best Practice Guidelines, 39th Aerospace Sciences Meeting and Exhibit, AIAA, 2000.

[22] M. Casey, and T. Wintergerste, *Best Practices Guidelines: ERCOFTAC Special Interest Group on Quality and Trust in Industrial CFD*, Ercoftac, 2000.

[23] A. Rizzi, J. Vos, Towards Establishing Credibility in Computational Fluid Dynamics, *AIAA Journal* **36**(5):668-675,1998.

[24] I. Celik, J. Li, G. Hu, C. Shaffer, Limitations of Richardson Extrapolation and Some Possible Remedies, *J Fluids Eng*, **127**:795-805, 2005.

[25] B. E. Launder, and D. B. Spalding, The Numerical Computation of Turbulent Flows, *Computer Methods in Applied Mechanics and Engineering*, vol. **3**:269-289, 1974.

[26] Y. S. Chen, and. S. W. Kim, *Computation of turbulent flows using an extended k-epsilon turbulence closure model*, NASA STI/Recon Technical Report N 88, 1987.

[27] D. S. Jang, R. Jetli, and S. Acharya, Comprension of the PISO, SIMPLER and SIMPLEC Algorithms for the Treatment of the Pressure-Velocity Coupling in Steady Flow Problems, *Num Heat Transf.* **10**.3:209-228, 1986.

[28] ASME Editorial Board, Journal of Heat Transfer Editorial Policy Statement on Numerical Accuracy, *ASME Journal of Heat Transfer*, **116**:797-798, 1994.

[29] H. W. Coleman, and F. Stern, Uncertainties and CFD Validation, *ASME Journal of Fluids Engineering*, **119**:795-803, 1997.

[30] P. J. Roache, Quantification of Uncertainty in Computational Fluid Dynamics, *Annual Review of Fluid Mechanics*, Vol. **29**:123-160, 1997.

[31] A. M. Morega, and A. Bejan, Heatline visualization of forced convection in porous media, *International journal of heat and fluid flow* **15**.1:42-47, 1994.

[32] A. Bejan, *Convection heat transfer*, John wiley & sons, 2013.

[33] A. S. Monin, A. M. Yaglom and J. L. Lumley, Mechanics of Turbulence, In *Statistical Fluid Mechanics*, **2**-vol. set, MIT Press, 1979.

[34] A. Aziz, A similarity solution for laminar thermal boundary layer over a flat plate with a convective surface boundary condition, *Communications in Nonlinear Science and Numerical Simulation*, **14**.4:1064-1068, 2009.