Research Article

Experimental investigation and numerical verification of Coanda effect on curved surfaces using co-flow thrust vectoring

Emre Kara [a],* and Hüdai Erpulat [a]

ªAeronautics and Aerospace Engineering Department, Gaziantep University, Faculty of Aeronautics and Aerospace, Gaziantep 27310, Turkey

ARTICLE INFO	ABSTRACT

In this study, a popular co-flow thrust vectoring system, which is superior to typical Coanda nozzles with one main jet, is examined experimentally and compared with 2D and 3D computational fluid dynamics results. High Speed Orienting Momentum with Enhanced Reversibility nozzle concept is the base design to proposed configuration which uses a control jet additional to the main jet for better and active enhancement on the flow vectoring and streamlined side-walls resulted in less flow blockage. This comparatively novel concept is utilized in an experimental setup to direct the thrust of aerial vehicles. The system includes two inlets (inlet1, inlet2) with different jet velocities and one pintle to separate and smoothly direct these jets and a converging-diverging nozzle to enclose these components. Experimental study is accomplished with four different configurations of inlet1 and inlet2 as 15 m/s and 10 m/s; 20 m/s and 10 m/s; 30 m/s and 10 m/s, and 45 m/s and 10 m/s, respectively. The tangential velocities on the curved surfaces are successfully measured utilizing a micro-manometer (Pitot tube) so that attachments/detachments of jets on the exit walls and deflection angles are calculated for each inlet velocities. The current experimental study also revealed that 3D assumption of computational fluid dynamics of Coanda effect is highly accurate and deflection angle results are not far from experimental results with the average deficit of only 5.44 %. As the result, 3D verification study resembles to experimental study in terms of deflection angles for all configurations.

1. Introduction

This paper involves an experimental investigation of the Coanda effect thrust vectoring method in addition to a discussion about its potential implementation on aviation industry considering both its advantages and disadvantages. High Speed Orienting Momentum with Enhanced Reversibility (HOMER) nozzle concept [1] is selected as the base design to the proposed configuration. HOMER design is selected since it uses a control jet additional to the main jet for better and active enhancement on the flow vectoring and streamlined side-walls resulted in less flow blockage. The results of the deflection effect on the Coanda surfaces are also given in this paper.

Although these propulsion systems are developed for combat aircrafts, there is also a significant opportunity to use them in the civil aviation sector. All those propulsion systems have advantages and disadvantages with respect to each other. In today's aviation world, the most commonly used thrust vectoring types are given in Figure 1. Among them Coanda methods [2, 3] are most suitable for new generation Vertical Take-Off and Landing/ Short Takeoff and Landing (VTOL/STOL) air vehicles, with their quick responses they can have a huge impact even on futuristic city transportation systems in the following decades.

In the field of aerial jet propulsion, it is crucial to correctly orient the thrust for take-off/landing accuracy and maneuverability. Almost all aeronautical vehicles rely on movement of aerodynamic control surfaces. Despite its reliability, control surfaces have plenty of heavy mechanisms.
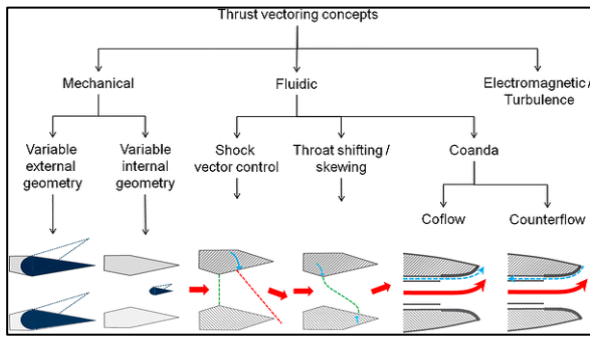
Figure 1. Thrust vectoring concepts [3]

In the last decade, the fluid thrust vectoring (FTV) concept is emerging as a popular choice [1-4], especially for the requirements of V/STOL aircrafts. The FTV method ensures the vectoring of the thrust without any movable mechanical parts simply and effectively. The affecting parameters of FTV are shortly classified as structure and flow parameters. Structure parameters include the nozzle geometry, location of the pintle tip, throat width. Flow parameters include the incoming jet(s), exit pressure, viscous flow over the smooth surface of the converging-diverging nozzle. In traditional nozzle systems, the flow has only one directed incoming jet and is oriented by the movement of exit diverging nozzle walls [4].

Coanda effect is the adhesion of a high speed jet to a convex surface. Coanda effect is essentially utilized in aerospace applications [5], heating/cooling [6] and marine technology [7]. There are three alternative nozzle configurations for the utilization of Coanda surfaces: Newman setup [4] with only one jet next to the convex surface, Juvet setup [8] with two primary jets and a control jet and HOMER nozzle setup [1] composed of two inlets, two Coanda surfaces, one pintle to separate and smoothly direct these jets and a converging-diverging nozzle to enclose whole components. In the patented study of 2D nozzle geometry, this nozzle concept is developed by Trancossi et al. [9] who finished an EU project (ACHEON) [1] regarding this exciting design. In the current study, it is aimed to develop a FTV concept by using similar Coanda surfaces. HOMER nozzle concept [1] is selected as the base design to the proposed configuration since it uses a control jet additional to the main jet for better and active enhancement on the flow vectoring and streamlined side-walls resulted in less flow blockage. Co-flow FTV is mostly used one in subsonic flows having one constant velocity and other jet stream with a controllable velocity is given as red and blue streams, respectively, as shown in Figure 2. This differential velocity allows the thrust vector slope becoming more controllable.

Although general purpose fluidic thrust vectoring is utilized for enhancing thrust maneuverability since late 1960s [10], the HOMER design is highly immature and open to improvement with its only 10 years of background. In this paragraph, the most recent article/project of the study groups working on HOMER design is taken into account. Starting from the original design of Trancossi et al. [1], Subash and Dumas [11] represented different turbulence models applied on air-jet flow tangential to a curved surface. They realized that jet deflection angle and the thrust are important parameters that should be handled with extreme care. They do not present an experimental study to compare and validate their CFD results. Another study group, Cen et al. [12] tried to integrate HOMER design to Unmanned Aerial Vehicle (UAV) for enhancing its maneuverability. They focused on developing this design starting from CFD simulation of the design, they proposed an integrated flight/thrust vectoring (HOMER) control scheme for fixed wing UAVs. They stated the effectiveness of the HOMER design adding to four-cascaded nonlinear dynamic inversion (NDI) control law they proposed. However, they do not build an experimental setup as in the previous reference to validate their results. In 2016, Trancossi et al. [13] generated the mathematical model of HOMER nozzle. Although the mathematical model of 2D case of the system is developed in the article, it can be counted as a step stone for future 3D mathematical modeling attempts. They validate the model by comparing the results with CFD simulations of velocity profiles and pressure drops through the selected sections. They found a good correlation of those outcomes despite frictional effect underestimation. In the most recent study, Panneer and Thiyagu [14] analyzed effect of pintle geometry on HOMER design. They found that sharper tip of a pintle can result in an increase on deflection angle. Additionally, the velocity plots are shown for different velocity ratio configurations. Although this newest study shows a new perspective on HOMER nozzle effectiveness, CFD results are not enough to confirm the HOMER nozzle concept and a new viewpoint should be developed both in experiments and numerical approaches. In the current study, both 3D CFD study and construction of a new HOMER nozzle design, experimental study and the verification of the results using the CFD study are all served to literature as one original complementary package.

HOMER nozzle concept [1] basically directs the jet direction as velocity of the main and control jet streams changes and clearance between the upper and lower walls of Coanda surface optimized. This configuration enhances the aircraft thrust system in following steps:

- Thrust vectoring capability increases with avoided stall on the diverging section of each wall.
- No-mechanical-part in movement results in reduction in total weight and total cost of structure.
- Quick and directional control of propulsion may be used as an advantageous maneuverability of V/STOL

aircrafts.

- Using more efficient control system on thrust vectoring means less use of fossil-based fuel.

2. CFD Study

In the previously published study [15], commercially available CFD software, ANSYS 17.0, was used as the mesh generator (ANSYS Meshing) and the flow solver (ANSYS Fluent). The boundary conditions for the solution domain are shortly depicted in Figure 3 both for previous 2D study and the currently done 3D study. Same parameters are employed for 3D study to compare them independently so that one can refer to previous study.

As suggested in [11], velocity ratio (VR = V2/V1) between channels inlet1 and inlet2 were selected as ranging from 1.5 to 3.0, i.e. 15 m/s and 10 m/s; 20 m/s and 10 m/s; 30 m/s and 10 m/s, respectively. In order to understand whether the upper limit of the velocity ratio is correctly stated as 3.0 in reference [11], a new case study with inlet channel velocities of 45 m/s and 10 m/s is also studied. Details can be found in Kara and Erpulat [15]. In the current experimental study, this previously worked CFD results are also referenced and compared at the end of the next section in addition to 3D CFD study that is shown to be more effective in the prediction of deflection angle over the Coanda surface.

As seen in Figure 4, four different case studies are selected for working fluid (ideal gas as air) with inlet1 (constant) velocity of 10 m/s and four different inlet2 (control) velocities of 15, 20 , 30 and 45 m/s for case study 1, case study 2, case study 3 and case study 4, respectively. Deflection angle is increased from 50° to a physical limit of 180° when VR increased from 1.5 to 4.5 respectively. These results are compared with experimental study and are also discussed in the proceeding section.

3. Experimental Study

The experimental setup consists of two Coanda surfaces, one pintle, two convergent ducts to direct the flow through the fans as shown in Figure 5. All the designed parts are produced utilizing 3D printer with PLA+ material.
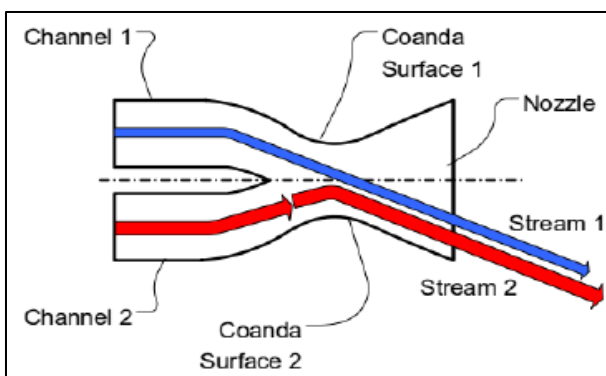

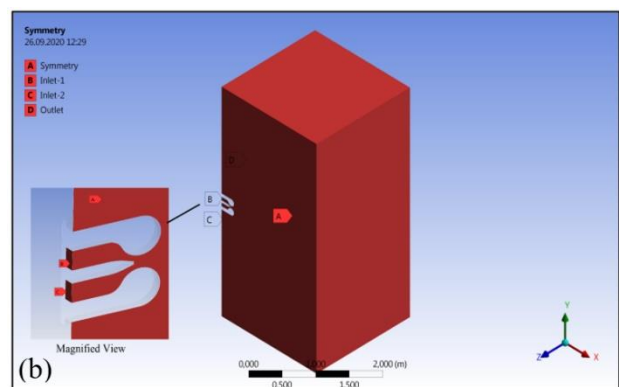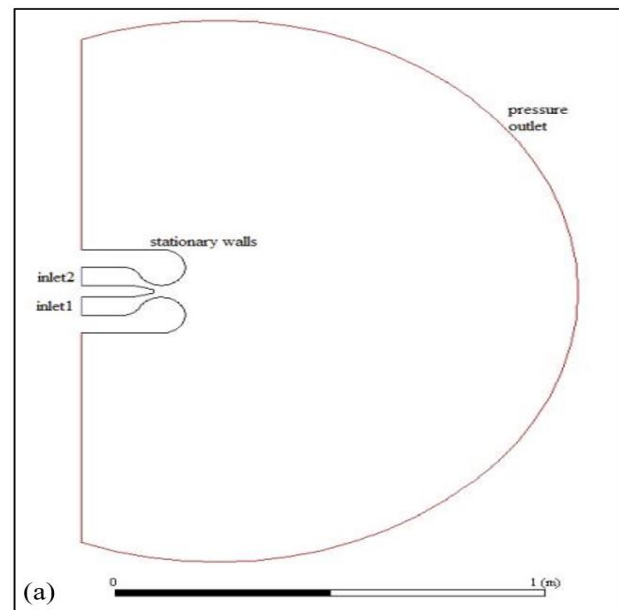Figure 2. Schematic of the HOMER nozzle [2]

Figure 3. Boundary conditions (velocity inlets, pressure outlet, symmetry plane) of suggested FTV design (a) 2D, (b) 3D

Application page of the 3D printer is captured and is shown in Figure 6. Propellers with ten blades are preferred due to the rotational speed of the selected motor is not enough for less blade propeller as empirically tested several times. DC motor is selected with 1400 kV, 27.8 mm x 27.8 mm duct area and 3.17 mm shaft diameter.

Plexiglass is used as cover material on the side surfaces as shown in Figure 5(a). This material is quite cheap and easy to manage. Coanda surfaces are completely covered with plexiglass. The ducted fans are combined appropriately by plexiglass support. The inner corner of the Coanda surfaces is attached to the side surfaces by means of screws. This connection allows Coanda surfaces to move but remain in the desired position. The transducer surfaces are fastened to the Coanda surfaces by side screws. Motor cables are pulled out from the holes which drilled on the circle to rectangle converter and connected to the motor drives. PCE-PFM 2 brand Micro Manometer with Pitot tube (PCE Instruments, Southampton, UK) is employed for accurately measuring air flow speed in m/s with PCE software for data processing. List of equipment mentioned here are summarized in Table 1 with their working specifications.
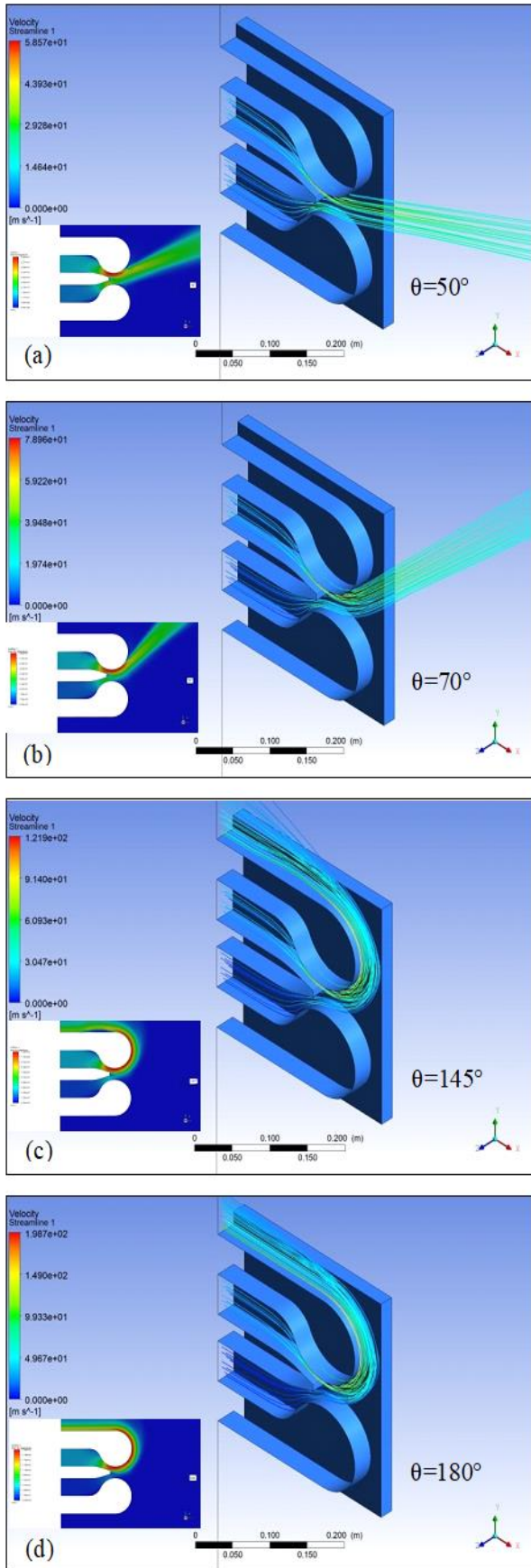
Figure 4. 3D streamlines and velocity contours projected onto symmetry surface for VR of (a) 1.5, (b) 2.0, (c) 3.0 and (d) 4.5 with deflection angles (θ) next to them.
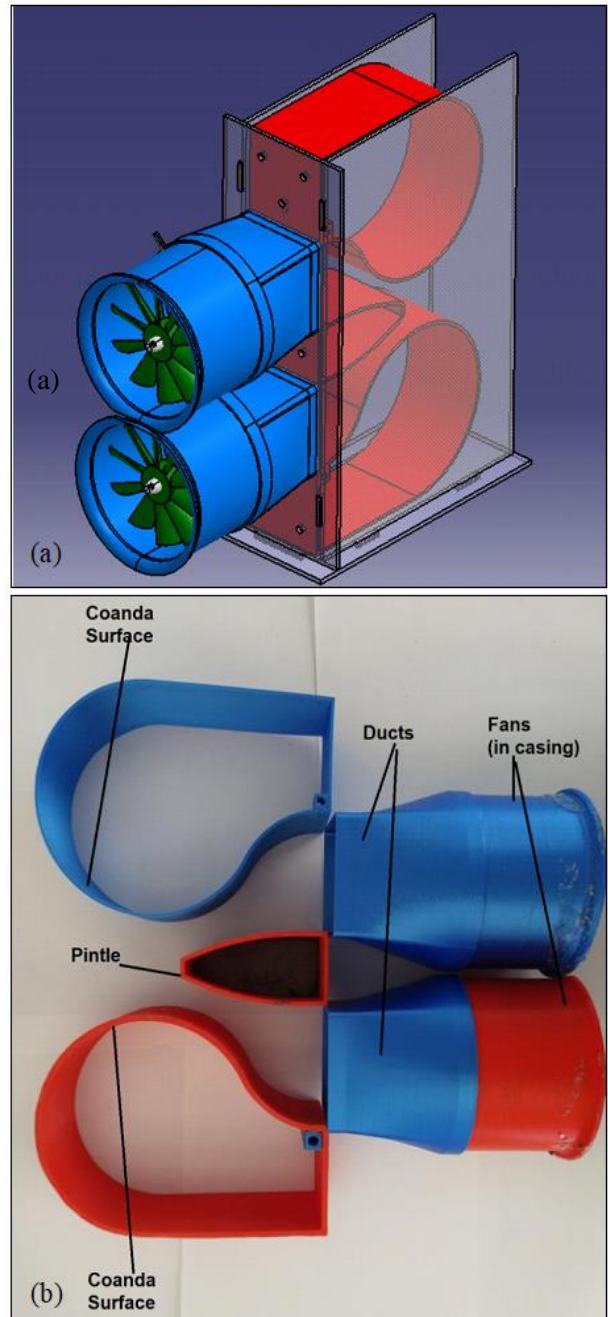

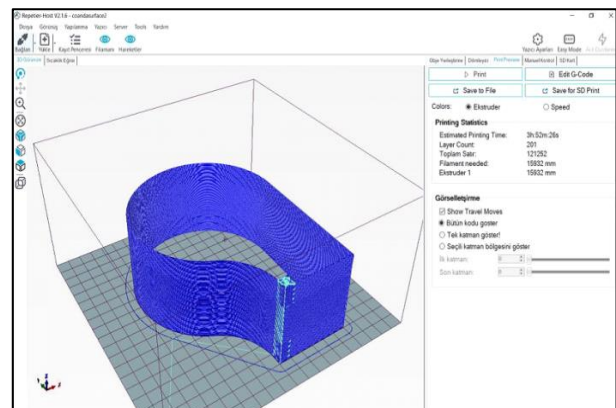Figure 5. (a) 3D model of the experimental setup and (b) the constructed experimental setup with components explained

Figure 6. Repetier-host software snapshot of utilized 3D printer

Table 1. Used equipment and their working specifications

| Brand name	Equipment name (quantity)	Specifications
PCE-PFM 2	Micro manometer	For pressure: M. ranges: 50 mbar Resolution: 0.01 mbar Uncertainty: ±0.3 % of the measuring range Max. pressure: 10 psi Response time: 0.5 seconds For air velocity: Measurement ranges: 1…80 m/s Resolution: 0.01 m/s Uncertainty: ±2.5 % of the measuring range Dimensions: Unit: 210 x 75 x 50 mm Pitot tube: 375 x 205 mm Weight: 280 g
Wester A2212	DC motor (2)	Brushless outrunner motor: Shaft diameter: 3.17 mm Dimensions: 27.8 mm x 27.8 mm Recommended Propeller: 2-cells 7 x 3.5 or 3-cells 6x3 Weight: 48g
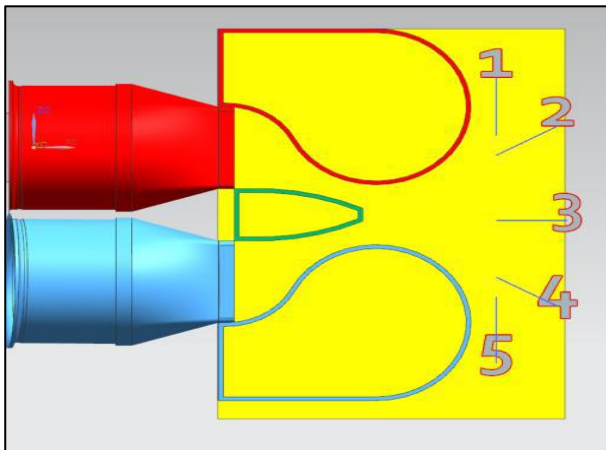
Figure 7. Velocity measurement sections for micro manometer with Pitot tube

Table 2. Velocity Ratio versus sections of velocity measurements over the Coanda surfaces

VR	Section 1	Section 2	Section 3	Section 4	Section 5
1.0*	2.1 m/s	4.3 m/s	12.2m/s	4.3 m/s	2.1 m/s
1.5	6.7 m/s	7.1 m/s	6.0 m/s	-1.2m/s	-0.4 m/s
2.0	9.9 m/s	8.2 m/s	6.9 m/s	-1.8m/s	-0.8 m/s
3.0	9.2 m/s	4.8 m/s	3.0 m/s	-2.0m/s	-1.0 m/s
4.5	13.1m/s	6.1 m/s	3.4 m/s	-2.8m/s	-1.2 m/s

Baseline control test case

The measured data are transferred to a computer with the included USB adapter and software. Measurement locations are shown in Figure 7. Sections are separated 45° apart from each other and baseline is taken as section 3 (the horizontal section). All results are tabulated in Table 2.

As expected, the greater the difference between the electrical motors speeds means the more vectoring of the co-flow. Furthermore, during the execution of the experiments, it was observed that the Coanda effect is increased when the surfaces were brought closer to each other and the adhesion effect increased when the flow was forced through a narrower opening over the pintle. Considering experimental results of VR's for different sections in Table 2, curves are fitted for each set of experiments with R^2 of 0.96 or more. Thus, Table 3 summarizes the experimental outcomes of deflection angles at different VR's.

Comparing previously studied 2D CFD study [15] with the current experimental study reveals that 2D assumption of the fluid dynamics of Coanda effect is highly inaccurate. Moreover, it overshoots the deflection angle, θ, in low-to-moderate VR values and after VR > 2.0 it changes direction and underestimates the deflection angle, θ, at a total absolute average of 20.34 %. As seen in the 4th and 6th columns of Table 3, 3D CFD study is more accurate and is not far from experimental results with the absolute average deficit of only 5.44 %. This can be the result of secondary flows in third dimension and effect of side surface guidance which is not the case for a 2D study.

Authors understood that an assumption of no secondary flow and dominantly axisymmetric behavior of the flow are both incorrect. Thus, it is studied in three-dimensional computational domain with careful statement of the flow behavior in boundary layer and separation regions over the Coanda surfaces. Experimental outcomes of the current work are carefully examined and compared with 3D CFD study. In order to understand whether the upper limit of the velocity ratio is correctly stated as 3.0 in reference [11], a new case study with inlet channel velocities of 45 m/s and 10 m/s is also studied experimentally as shown in Table 2 and the limiting VR is found to be 4.5 in the experimental setup as in the cases of CFD studies.

Table 3. Velocity Ratio versus experimental deflection angles, θ, (with $R^2 \geq 0.96$) and numerical (CFD) deflection angles, θ

VR	θ (Exp)	θ (2D CFD)	θ (3D CFD)	Error of 2D CFD results [%]	Error of 3D CFD results [%]
1.0*	0.0	0	0	0.00	0.00
1.5	56.8	65	50	14.44	-11.97
2.0	66.4	80	70	20.48	5.42
3.0	134.0	95	145	-29.10	8.20
4.5	177.0	110	180	-37.85	1.69
Average error relative to experiments				20.37	5.44

Baseline control test case

4. Conclusion

In this study, a novel propulsion system is examined. It is understood how this kind of effort can change future of the thrust vectoring in VTOLs.

As one of the key outcomes, firstly, it is observed that the velocities of the air supplied from two different ducts are controlled free from each other and this concept is proved experimentally. Throughout the numerical study, the effect of geometry properties on mesh properties was examined in 3D. Particularly, the importance of pre-study of experiments was realized utilizing the desired quality of the mesh. Results revealed that the outcomes of the setup are not compatible with 2D CFD results, especially in the range. This is because two-dimensional approach for fluid dynamics of the Coanda effect is not adequate to correctly define the real flow as explained in the results section. Thus, the experimental results are compared with 3D CFD analysis considering secondary flows that are not possible with 2D approach. Results showed that 3D verification study resembles to experimental study in terms of deflection angles for all configurations.

The new insights that should be investigated and be revealed can be summarized as follows:

- Experimental setup should be improved after currently in-progress project so that powerful motor in ducted fans, superior materials for the cover without unwanted airflow leakages, etc.

- Visualization will be added as another improvement to the current experimental study employing Schlieren flow observation techniques.

- Better and more expensive choices will be investigated for flush-mounted shear stress sensors and static pressure measurement devices in order to increase the diversity of focused techniques. Thus, the behavior of the airflow over Coanda surfaces will be understood and inquired profoundly.

Declaration

The authors declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article. The authors also declared that this article is original, was prepared in accordance with international publication and research ethics, and ethical committee permission or any special permission is not required.

Author Contributions

E. Kara and H. Erpulat conceived the study together. E. Kara supervised the research, analyzed the data and drafted / finalized the paper. H. Erpulat conducted the experiments, contributed to the data analysis and revised the paper.

Nomenclature

θ	: Deflection Angle [°]
FTV	: Fluid Thrust Vectoring
CFD	: Computational Fluid Dynamics
HOMER	: High Speed Orienting Momentum with Enhanced Reversibility
NDI	: Nonlinear Dynamic Inversion
STOL	: Short Takeoff and Landing
UAV	: Unmanned Aerial Vehicle
VR	: Velocity Ratio
VTOL	: Vertical Take-Off and Landing

References

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

2. Trancossi, M., A. Dumas, S. S. Das, and J. Pascoa, *Design methods of Coanda effect nozzle with two streams.* Incas Bulletin, 2014. **6**(1): p. 83-95.

3. Bougas, L., and M. Hornung, *Propulsion system integration and thrust vectoring aspects for scaled jet UAVs.* CEAS Aeronautical Journal, 2013. **4**(3): p. 327–343.

4. Newman, B. G., *The Deflexion of Plane Jet by Adjacent Boundaries Coanda Effect.* 1961, UK: Pergamon Press.

5. Jain, S., S. Roy, D. Gupta, V. Kumar, and N. Kumar, *Study on fluidic thrust vectoring techniques for application in V/STOL aircrafts.* SAE Technical Paper, 2015. No. 2015-01-2423.

6. Sidiropoulos, V., and J Vlachopoulos, *An investigation of Venturi and Coanda effects in blown film cooling.* International Polymer Processing, 2000. **15**(1): p. 40-45.

7. El Halal, Y., C. H. Marques, L. A. Rocha, L. A. Isoldi, R. D. L. Lemos, C. Fragassa, and E. D. dos Santos, *Numerical study of turbulent air and water flows in a nozzle based on the Coanda effect.* Journal of Marine Science and Engineering, 2019. **7**(2): 21.

8. Juvet, P. J. D., *Control of high Reynolds number round jets*, in Mechanical Engineering 1993, Stanford University: USA, TF-59.

9. Trancossi, M., A. Dumas, I. Giuliani, and I. Baffigi, *Ugello Capace di Deviare in Modo Dinamico e Controllabile un getto Sintetico senza parti Meccaniche in Movimento e suo Sistema di Con trollo*, 2011, Patent No. RE2011A000049, Italy.

10. Springer, A., *50 Years of NASA Aeronautics Achievements.* 46th AIAA Aerospace Sciences Meeting and Exhibit , p. 859.

11. Subhash, M., and A. Dumas, *Computational study of Coanda adhesion over curved surface.* SAE International Journal of Aerospace, 2013. **6**(2013-01-2302): p. 260-272.

12. Cen, Z., T. Smith, P. Stewart, and J. Stewart, *Integrated flight/thrust vectoring control for jet-powered unmanned aerial vehicles with ACHEON propulsion.* Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, 2015. **229**(6): p. 1057-1075.

13. Trancossi, M., J. Stewart, S. Maharshi and D. Angeli, *Mathematical model of a constructal Coanda effect nozzle.* Journal of Applied Fluid Mechanics, 2016. **9**(6): p. 2813-2822.

14. Panneer, M., and R. Thiyagu, *Design and analysis of Coanda effect nozzle with two independent streams.* International Journal of Ambient Energy, 2020. **41**(8): p.

851-860.

15. Kara E., and H. E., *Numerical Investigation of Jet Orientation Using Co-Flow Thrust Vectoring with Coanda Effect*, in ICAME2019: İstanbul, p. 1-8.