The Effect of Curvature Ratio on Flow Structureand Fluids Mixing in 90º bent square duct.

Abdul Satar Jawad Mohammed

Mechanical Engineering Department, University of Technology, Baghdad

asj_2006saraf@yahoo.com

Sadiq Geham Abed-Alfathel

Mechanical Engineering Department, University of Technology, Baghdad

saduqalshibly@yahoo.com

Submission date:- 31/12/2019	Acceptance date:- 11/10/2020	Publication date:- 27/10/2020

Abstract

Many mechanical and chemical problems rely mainly on the mixing characteristics of a dispersed liquid and the carrier gas which is strongly affected by the rates of mass and heat exchanged. The secondary flow generated by the centrifugal forces accompany a substantial change in flow direction leads to the presence of counter rotating vortices. The study investigates the effect of curvature ratio on the flow structure and turbulence intensity during a liquid-gas mixing process prior to a bent duct. The study employs the experimental Particle Image Velocimetry technique (PIV) in purpose tracking the secondary flow structure when the water-air mixture travelling through a bent duct. The curvature ratios were taken to be (0.25, 0.5, 0.75) at average velocities of 2.5 and 5m/s for air flowing through a square duct. The PIV images illustrate the appearance of a Pair of rotating Dean vortices (four-cell pattern) generated for all curvature ratios with the vortices near the inner side of the bend moved outward while decreasing the curvature ratio as a result of centrifugal effect and flow separation. The design and the configuration of the water nozzles matrix is decided according to the numerical simulation using ANSYS FLUENT 19.R1, with RNG-k-ε turbulent model. The numerical analysis showed that the swirl intensity has little effect on mixing due to changing Reynolds number and was more influenced by the changing of the curvature ratio. The phenomenal comparison between experimental and numerical results showed good agreement as the maximum deviation recorded is about (7.1%).

Keywords: Curved duct ;Curvature ratio; Flow structure ; (PIVLAB) Technique; Fluid mixing.

1-introduction

A large and important class of engineering problems is represented by internal viscous flows in curved duct. The flow through a curved tube has attracted considerable attention not only because of its practical importance in chemical and mechanical engineering, but also because of the interestingly physical features under the influence of the imbalance of centrifugal force and radial pressure. These features cause secondary motions in the form of counter rotating vortices. Internal flow through a duct for aircraft intakes, combustors, internal cooling system of gas turbines, ventilation ducts, wind tunnels etc. are the main application area of such ducts [1].Secondary flow is a pair of counter-rotating vortex cells called the Dean vortices showed by the mathematical analysis at the onset of these vortices in the curved channels resulting from the interaction between centrifugal forces and viscosity forces. If Dean number ($Dn = Re\sqrt{D_h/\breve{R}_c}$, where $\breve{R}_c = (R_C + D_h/2)$ is not large enough, the additional vortices will not form and the low structure will remain in two-vortex pattern due to small centrifugal force. However, when Dean Number is increased, another pair of counter-rotating will appear at a certain angle of bending [2].

The swirl intensity at the exit of the bend is a strong function of the bend curvature ratio (i.e. $\delta = R_c/D_h$) and a weak function of Reynolds number, and the normalized swirl intensity increased with the increase in elbow angle. When the curvature ratio of the bend is greater than 1.5, the secondary flow consisting of a pair of counter-rotating vortices is generated. At the same time, the velocity profile of the primary stream-wise flow is distorted and shifted away from the center of the curvature of the elbow. If R_c/D is smaller than 1.5, the flow becomes unsteady because of a flow separation occurring immediately downstream of the bend [3].

The flow separation can be clearly observed in the bending channel with strong curvature ratio where secondary velocity distributions are clearly stimulated by fluid movement from the inner wall to the outer wall of the curvature. However, no flow separation has been observed due to moderate curvature ratio [4]. The separation appears along the inner core with a reduction in curvature ratio (strong curvature), yet, at moderate curvature ratio (δ=1.5) no flow separation is observed [5]. When analytical solution in curved channels with rectangular cross-section was conducted, the result showed that the secondary flows in curved ducts appear as a pair of counter rotating vortices in small enough Dean Numbers. However, in large enough Dean numbers, instability is occurred in the flow field and new pairs of counter rotating vortices appear near the outer wall [6]. Three experimental techniques (PIV, LDV, Hot/Cold-Wire anemometry (HWA/CWA) were conducted on complex turbulent flows in a curved pipe at an angle of 90° with or without extra motion. The objective is to verify these complex flows in detail in terms of statistical quantities as well as the vorticial structures. The results showed that the PIV provides great possibilities for the study of structures, but still limited in statistics, while combined HWA/CWA was used to statistically analyze the flow field. LDA can be used to conduct any further investigation of some of the results for the two technologies above [7].

The theoretical study of fluid flow in a 90-degree curved pipe using the Detached Eddy Simulation (DES) turbulence model was employed to investigate the fluid flows at the Reynolds number range from 5000 to 20000. It turns out that as curvature ratio decreases the boundary layer separation becomes more obvious after 60°. In addition, the vortex appears clearly near the outer side of the center section of the curved pip end due to the separation zone and centrifugal force, where the boundary layer separation zone is expanded at small curvature ratio making the internal flow even more disordered until it reaching to the maximum value at 90° [8].

The water experiments with two types of elbows at different curvature ratios (δ=0.5, δ=1) were conducted in order to investigate the interaction between flow separation and the secondary flow due to the elbow curvature, under Reynolds number range from (1.8 to 5.4×105). The velocity fields in the elbows were measured using a high-speed Particle Image Velocimetry (PIV) founding that the flow separation always occurred in the lower curvature ratio (δ=0.5) while the flow separation occurred intermittently in the higher curvature ratio case (δ=1) [9]. A numerical study was conducted using previous experimental data and the assessment showed that the RNG k-ε turbulence model gives good results for primary stream-wise velocity and secondary swirling velocity profiles compared to other turbulence models [10]. The effect of curvature ratio on the mixing process involved in evaporative cooling of air by water spray is conducted for wind tunnel with 50 cm square cross-sectional area by using three different curvature ratios of 0.25, 0.5 and 0.75. The results showed that the higher curvature ratio of 0.75 gives the best performance of the humidification system due to the shrink in the separation zone accompany the weakening of centrifugal forces. This effect would widen the mixing area for the two streams to react and enhancing the mass and heat transfer rates between the water droplet and the surrounding air producing better cooling and humidification [11].

The motivation of the present work was to explore secondary flow patterns through Particle Image Velocimetry (PIV), to study the effect of the curvature ratio of the bent duct and Re number on the flow structure generated. The improvement of fluids mixing downstream the bend is to be decided according to the best curvature ratio that achieves the better humidification of air stream in practical situation such as the inlet ducting of a gas turbine generating unit aiming to moisten the air and reduce its temperature in purpose of enhancing the power output during hot summer climates.

2- Experimental Setup

2.1 Wind tunnel

The test rig used in this study is mainly a subsonic wind tunnel has a square cross section with 50 cm sides. The layout of the wind tunnel is given in figure (1). The air enters the wind tunnel through a bell mouth shaped duct that aims to reduce the effects of inlet turbulence and produce steadily flowing air. The bell mouth is connected to the first 3 m straight duct to accommodate the presence of inlet air preheaters and humidifier. The bent portion is connected to the end of the first straight duct to generate the secondary flow needed to help mixing the injected water with the air stream. In the present work, three bent ducts were used with curvature ratios of 0.25, 0.5 and 0.75 as shown schematically in figure (2). These curvature ratios ($\delta = R_c/D_h$) are chosen according to the regulations of ASHRAE [12]. The downstream end of the bent duct is connected to another 3 m straight duct leading the air to the axial fan running at 1500 (rev/min), which is responsible for the induction of air through the wind tunnel.

The airflow is adjusted by double butterfly gates built into the fan outlet. The air mean velocity at fully opened gates is 5 m/s, (Re = $1.43*10^5$), while at half opened gates is 2.5 m/s, (Re = 7.14 *104).

The air inlet condition is kept fixed throughout all tests at 45 C° dry bulb temperature (Td) and 15% relative humidity by adjusting electric preheaters and a steam humidifier. The properties of the humid air (temperature and relative humidity) are measured using a 25 sensors type (DHTD-22) at the main test section installed at the fan entrance, as mentioned in figure (1) above. The aim of this system is to achieve the necessary cooling by introducing small water mist into the air inside the curved duct with different curvature ratio using the nozzles matrix containing nine nozzles with a 0.1mm diameter placed in the first straight duct with the ability to be moved axially in advance to the bent duct inlet and also the nozzles are capable to rotate from – 90o to 90o with the axial flow direction, The matrix is connected to the fog machine supplying water under a pressure of 70 bar, as shown in figure (3). The boundary conditions of air and water adopted in the current study are listed in Table (1).

2-1 Digital Particle Image Velocimetry in MATLAB (PIVlab)

Flow structure was measured downstream the bent duct to study the effect of changing curvature ratio on the mixing of injected water droplets and the air due to the generated secondary flow. The Particle image velocimetry in MATLAB (PIVlab) was used as a non-intrusive analysis technique, where the PIV system consistence from the seeding particles system. The laser source and high speed camera were used to illuminate the flow structure through holding images of the flow, see figure (4). The particles seeding system is used to introduce the particles into the air stream in order to simulate the suspension of water droplets carried by air stream. The particles carried by the main air are illuminated to facilitate holding images at the section downstream of the curved part where the laser beam is supplied. For the purpose of stirring the particles in the seeder, a stream of compressed air at a pressure of 3 bar is supplied to the seeder from compressor. The seeder is connected with particle distributor to ensure that the particles are evenly distributed. The seeder consists of five tubes those distribute the particles in impact pin manner to expand the propagation diameter. The laser source (laser diode 532 nm green wave lengths and 180 mw) was installed at the end of the bent duct to generate a laser sheet with 2 mm thickness. The high speed camera was installed at the same location in a hatch made to depict the structure of the secondary flow downstream the bent duct.

The Particle image velocimetry in MATLAB (PIVlab) is used. A GUI-based open-source tool (PIVlab) for DPIV analyses in MATLAB is presented and developed by Thielicke, W. (2014) [13]. The image processing program depends on particle displacement by evaluating the cross-correlation of many small sub-images (interrogation areas). Through the drawing of particle travelled paths from image A to B images ,thus determining the flow structure, where the image experimental results (PIVLAB-technique) will be compared with numerical model in (ANSYS fluent).

3 – Numerical simulation

The numerical analysis in current study aims to study the effect of curvature ratio on the flow structure and mixing enhancement of two phase liquid-gas streams. The analysis steps of the FLUENT package were used to develop the CFD model of ANSYS FLUENT 19.R1. The RNG k-epsilon turbulent model as compared with the rest of the disturbance models is found more accurate for areas where a turbulent flow passes through bend channels [10] that correspond to our present work, the wind tunnel geometry simulation and boundary conditions are shown in Figure (5). The constants adopted in the program are listed in Table (2).

The assumptions adopted in our current study are listed as follows:

- Steady state.
- The air continuous flow is treated as single phase homogeneous.
- Water is treated as discrete phase flow.
- Incompressible. Turbulent flow.
- Three-dimensional.
- Newtonian fluid.

4 –Results and discussion

The images taken experimentally were processed by (PIVLAB) technique to determine the nature of the flow structures after the bent duct with two curvature ratio (0.25, 0.5) and numerically using the FLUENT package to validate the flow structure which has been extracted by the (PIVLAB) technique. In addition the flow structure through a bend with curvature ratio (0.75) was investigated only by numerical analysis. The flow conditions adopted in the current study are listed in Table (3). These flow conditions are selected to study the effect of increasing Reynolds number on the flow structure and air mixing at different curvature ratio of the bend duct.

All flow conditions showed the presence of a pair of rotating vortices (four-cell pattern) , and that the first pair of vortices be near the outer wall of the bend and the second pair of vortices close to the inner side of the bend , as indicated in figure (6,7,8, 9). For the case of bent duct with low curvature ratio, as shown in figures (6 and 7), the second pair of vortices is slightly moved toward the outer wall of the bend as a result of the flow separation due to the high centrifugal forces. Changing the fluid velocity with the same curvature ratio has no effect on the shape of the flow structure and number of vortices. This is attributed to the fact that the secondary flow structure and velocity profile are essentially dependent on the centrifugal forces and the longitudinal pressure gradient across the duct both which are strongly affected by the radius of curvature. Whence, as this radius is unaltered, the flow structure through the bend remains the same and show no change with changing air velocity.

The numerical analysis gives the same number of vortex pairs for all flow conditions with acceptable agreement with the flow structures obtained experimentally, as estimated never exceeds 7.1%. The flow structure that only numerically investigated with curvature ratio (0.75) show that the second pair of vortices is close to the inner wall of the third bent duct due to the weakness of centrifugal forces, where the separation zone almost smaller, see figure (10).

The effect of the Reynolds number and the curvature ratio on the swirl intensity (defined in the study as: $Is = \Omega D_h/2U_b$, where Ω the angular velocity) [7] was examined with the induced mixing at the exit of the bend (Φ= 90). The comparison of results with previous experimental and numerical results shows that mixing by the swirl intensity has little affected by changing the Reynolds number and was more influenced by the changing of curvature ratio as shown in Figures (11) and (12).

In general, the results obtained experimentally and numerically showed that changing the flow velocity through the same bend is not the controlling role in developing the structure of the vortices generated, but the bend geometry is, i.e. the curvature ratio. The strong bending (low curvature ratio) leads to increase the centrifugal forces pushing the second pair of vortices towards the outer wall of the bent duct. On the other hand, the mixing process downstream the bend is governed by two factors, the swirl intensity and the area of mixing. Therefore, although the swirl intensity is weakened at low bending duct (higher curvature ratio), but produce good mixing between the liquid and the carrier gas due to increasing the area of mixing as result of the shrinking in the separation zone accompany the weakening of centrifugal forces.

5-The comparison with previous work

Figure (13) shows a qualitative comparison between the present work and a previous numerical simulation conducted by P. Dutta [4]. The explored results stand for the growth of the separation zone for the curvature ratios of (δ=0.75, δ=0.5, δ=0.25) in both simulations. The comparison reveals the similarity in general trend for both investigations, as decreasing the curvature ratio will increase the area of separation with the same extent and location of separation point in both works.

6-conclosion

The turbulent flow structure was tested experimentally and numerically downstream of a 90°curved square duct at different curvature ratios and Re number to estimate their effect on the liquid-gas mixing process, and the following conclusions were obtained:

- Two pair of rotating vortices (four-cell pattern) with the first pair of vortices being near the outer wall of the bend while the second pair close to the inner wall of the bend.
- The second pair of vortices for a bent duct with low curvature ratio is slightly moved toward the outer wall as a result of the flow separation and high centrifugal force.
- Changing the fluid velocity at the same curvature ratio has no effect on the shape of the flow structure and number of vortices.
- The mixing quality measured by the swirl intensity shows little effect by changing the Reynolds number, but more influenced by changing the curvature ratio.
- The numerical analysis showed good agreement with experimental results with maximum deviation of 7.1%.

Table (1) List of boundary conditions of air and water adopted in the current study.

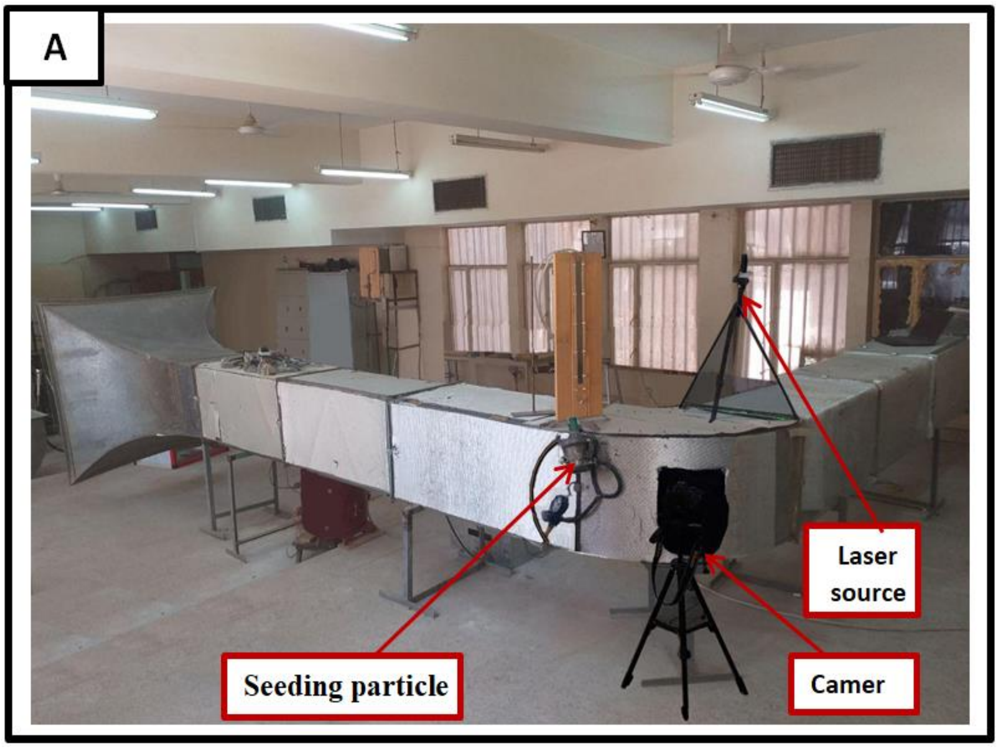
Air velocity (m/s)	Water flow rate (kg/h)	Td of air inlet (°C)	RH of air inlet (%)
5 and 2.5	19	45	15

Table (2) List of constants that adopted in the program.

<table>
<tr><th>Gravitational Acceleration In Y- direction</th><th>Model</th><th colspan="2">Optional model coefficients</th><th>Operating pressure</th></tr>
<tr><td rowspan="2">-9.81(m/s^2)</td><td rowspan="4">RNG K- epsilon</td><td>Cmu</td><td>0.0845</td><td rowspan="4">101325 (pascal)</td></tr>
<tr><td>C1-Epsilon</td><td>1.42</td></tr>
<tr><th>Turbulent intensity (%)</th><td>C2-Epsilon</td><td>1.68</td></tr>
<tr><td>3.7, 4</td><td>Wall Prandtl Number</td><td>0.85</td></tr>
<tr><th colspan="2">Air operating temp. inlet (K)</th><th colspan="3">Air mass fraction</th></tr>
<tr><td colspan="2">318</td><td colspan="3">From Species tab select the mass fraction air at RH = 15% as ($h_2o = 0.008996, o_2 = 0.21\%$)</td></tr>
</table>

Table (3) List of flow conditions investigated in the current study, at Td=45°C, RH=15%.

TEST NO.	U_b (m/s)	Re	δ	Dn	Investigated
Run 1	2.5	7.14*10^4	0.25	8.25*10^4	Experimentally + Numerically
Run 2	5	14.3*10^4	0.25	16.5*10^4	
Run 3	2.5	7.14*10^4	0.5	7.15*10^4	
Run 4	5	14.3*10^4	0.5	14.3*10^4	
Run 5	2.5	7.14*10^4	0.75	6.38*10^4	Numerically
Run 6	5	7.14*10^4	0.75	12.8*10^4	



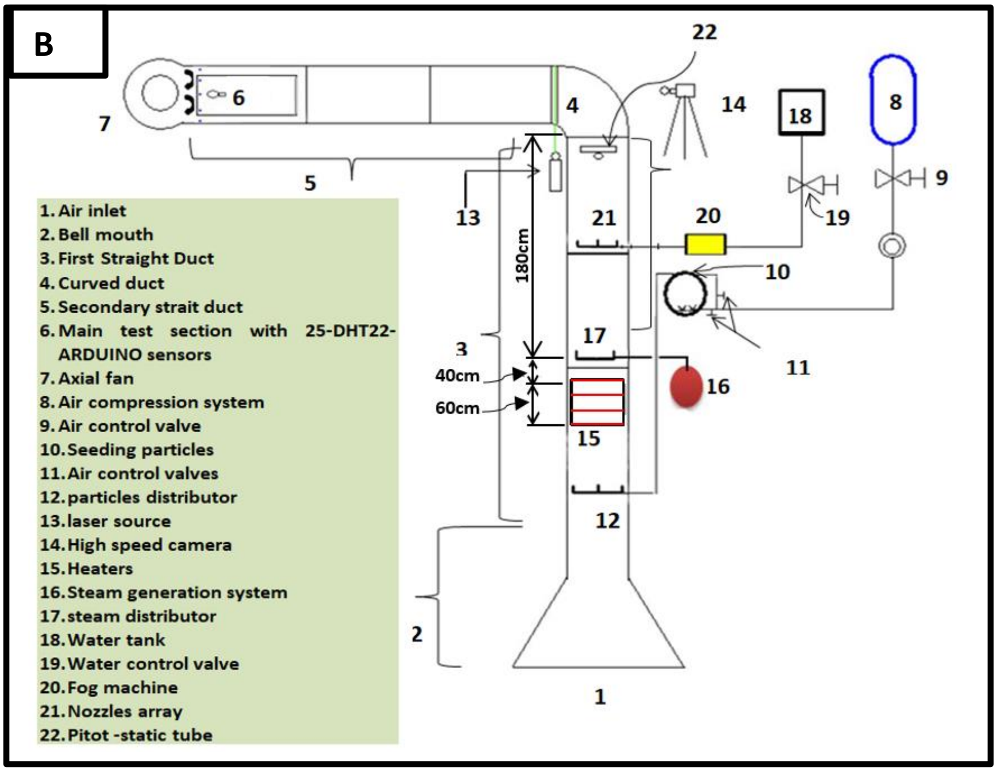


Figure (1): Layout of the wind tunnel (A) showing the camera hatch and seeding system in, (B) shows the diagram of the entire wind tunnel with the installation of air, water and (PIVLAB) systems

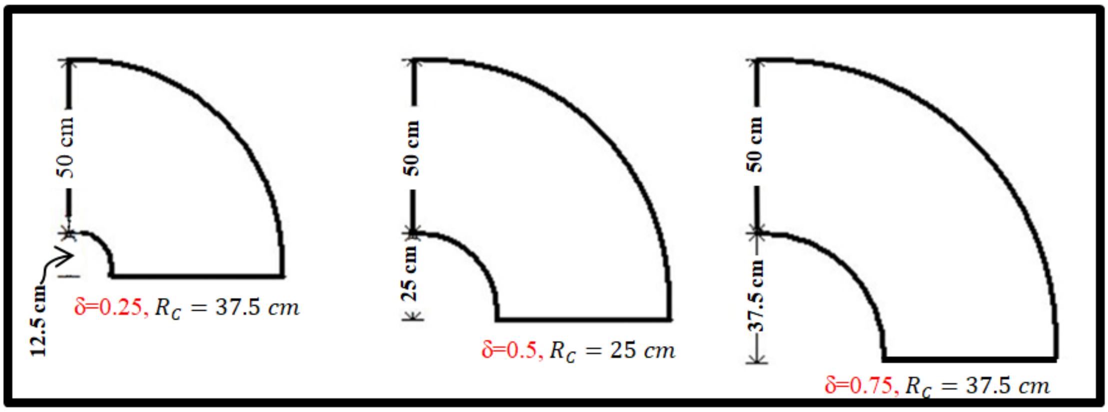


Figure (2): three bent ducts with curvature ratio (δ =0.25, δ =0.5, δ =0.75) and square cross section of (50*50) cm.

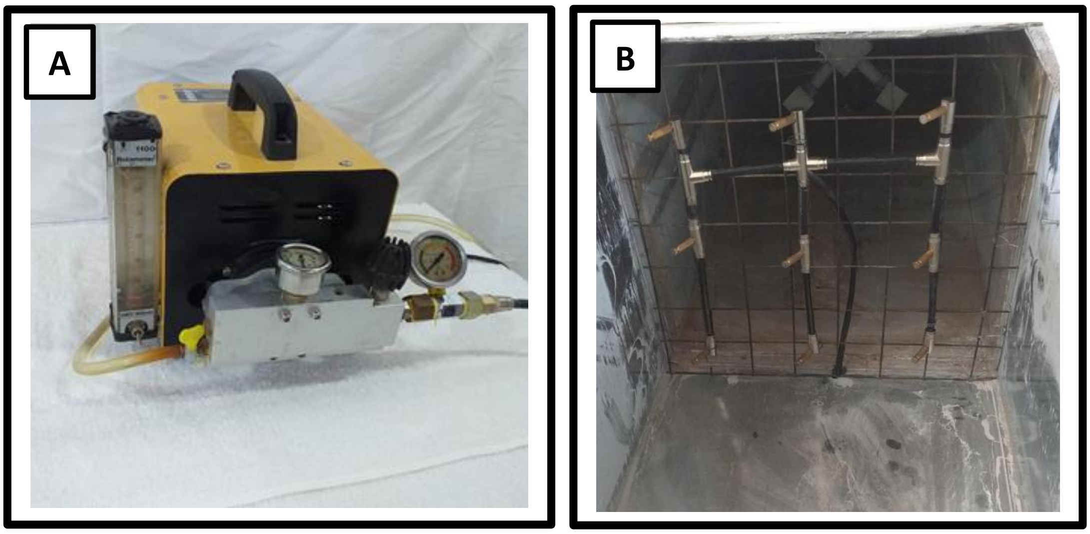


Figure (3) High pressure fog machine and accessories, (B) Nozzles matrix.

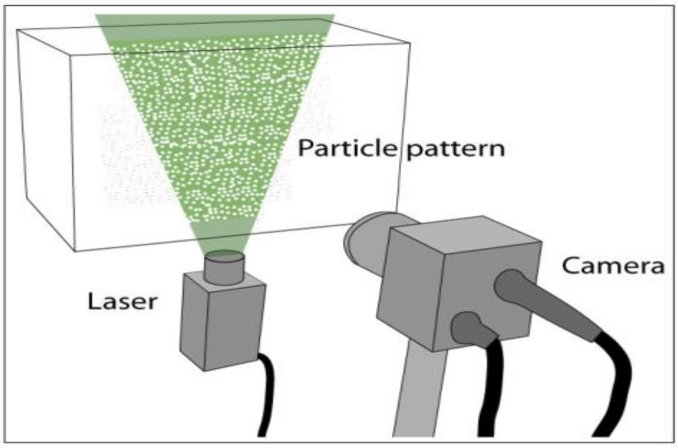


Figure (4): DPIV system arrangement showing the laser sheet illuminating the particles carried by the air flow.

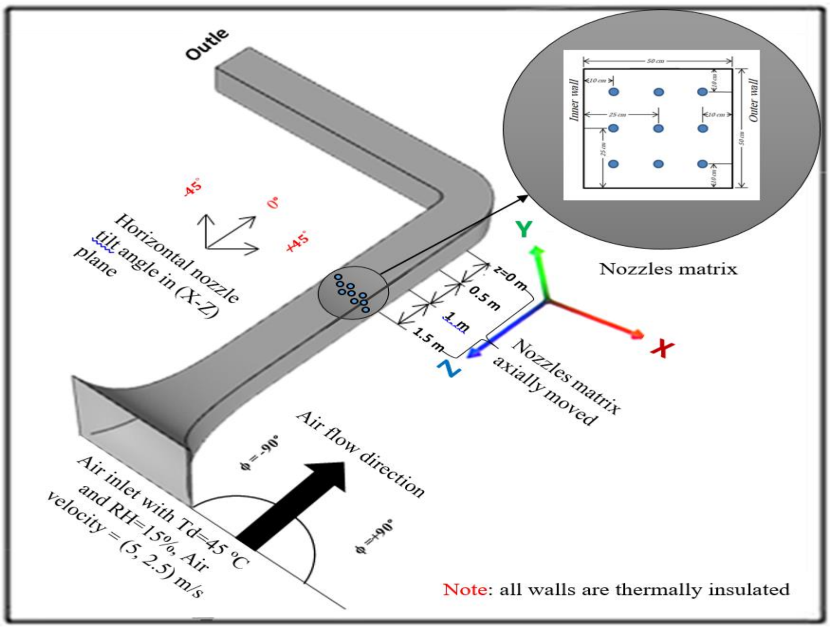


Figure (5): The wind tunnel geometry simulation with illustrated the boundary conditions.

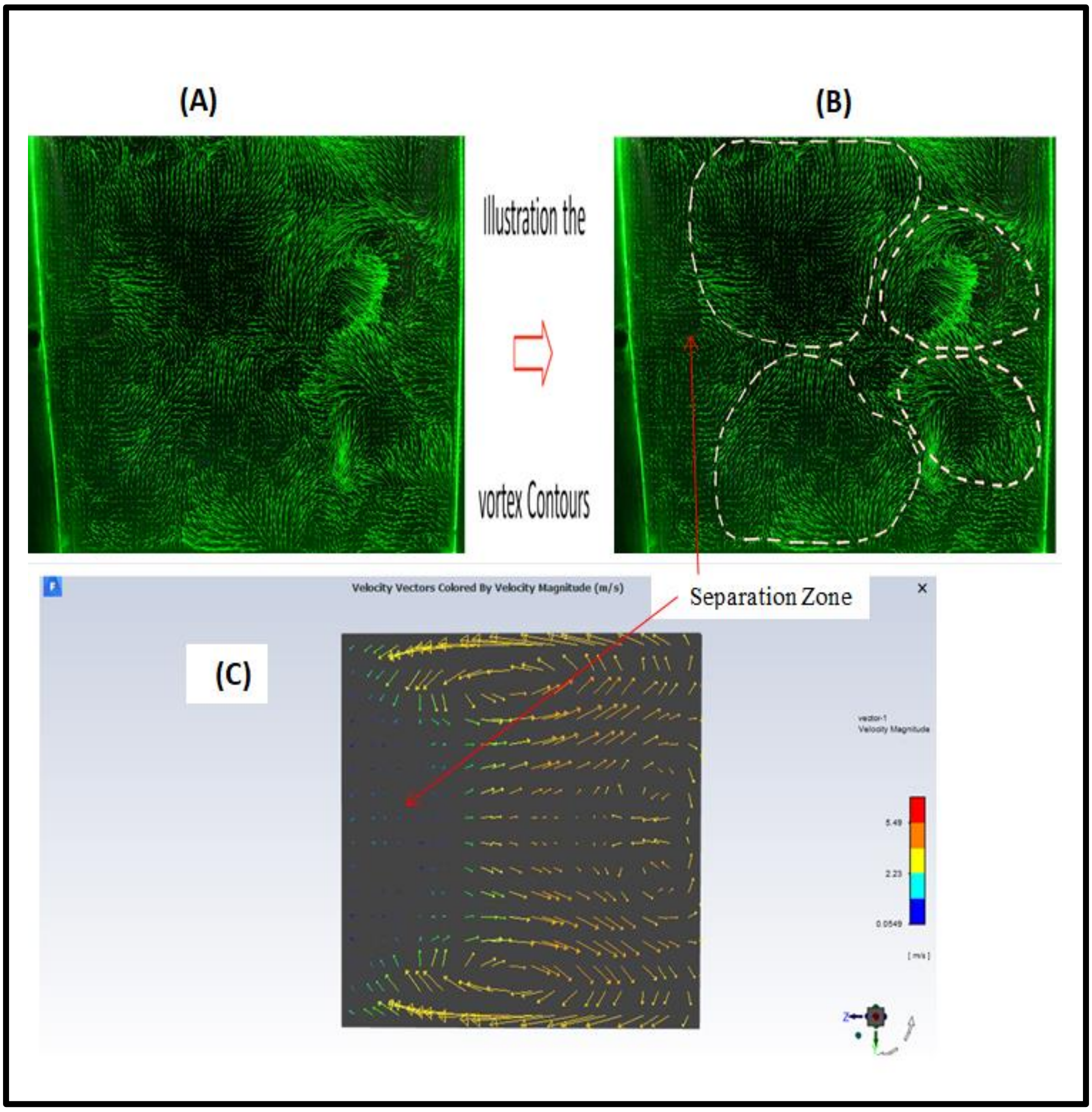


Figure (6): (A and B) Snapshots of the flow structure downstream the 90◦ bent duct, by (PIVLAB) measurement with curvature ratio δ=0.25 and test No.(Run 1), (C) flow structure at the same boundary condition obtained by ANSYS Fluent software simulation.

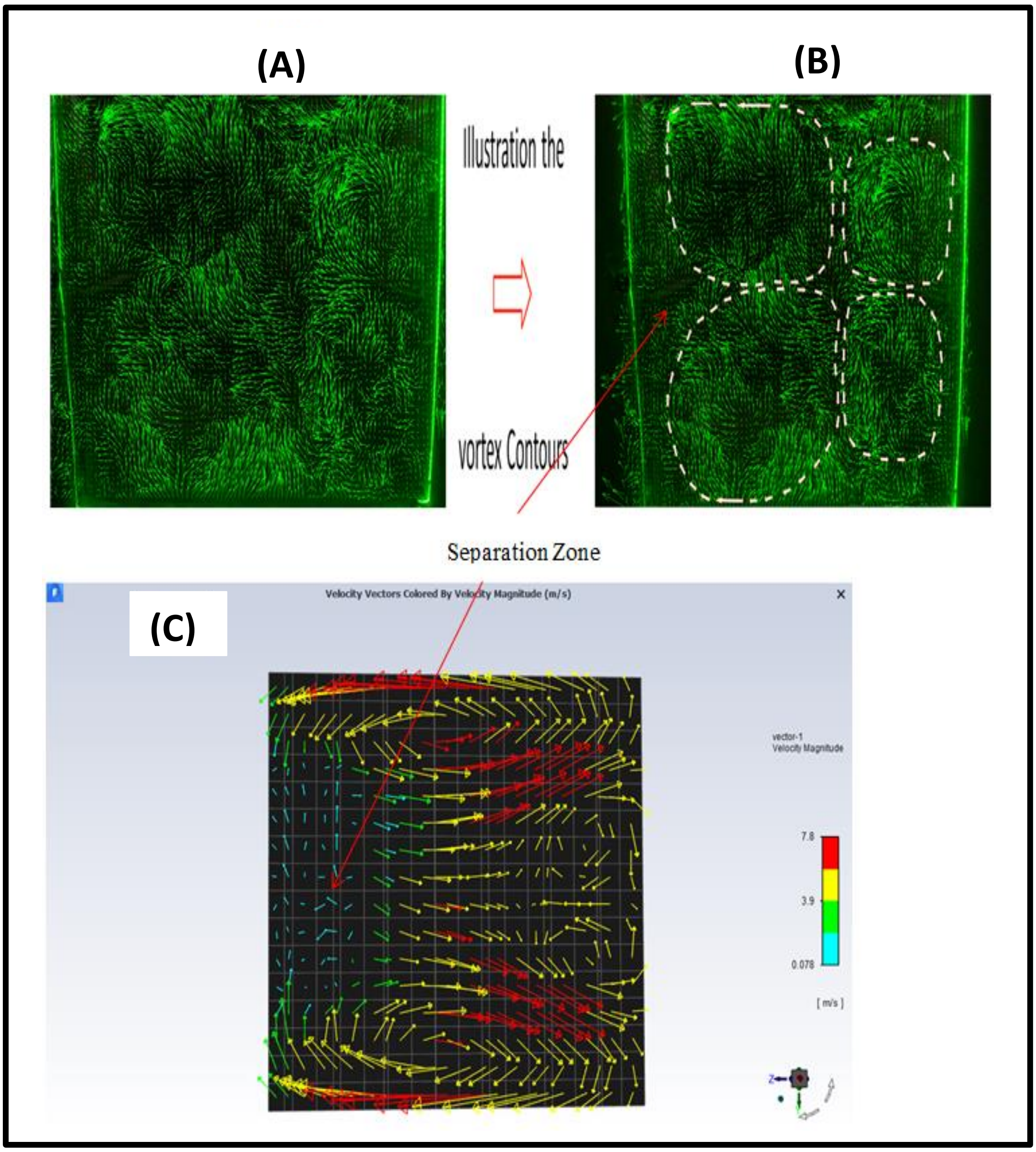


Figure (7): (A and B) Snapshots of the flow structure downstream the 90◦ bent duct, by PIVLAB measurements, with curvature ratio δ=0.25 and test No.(Run2), (C) flow structure at the same boundary condition obtained by ANSYS Fluent software simulation.

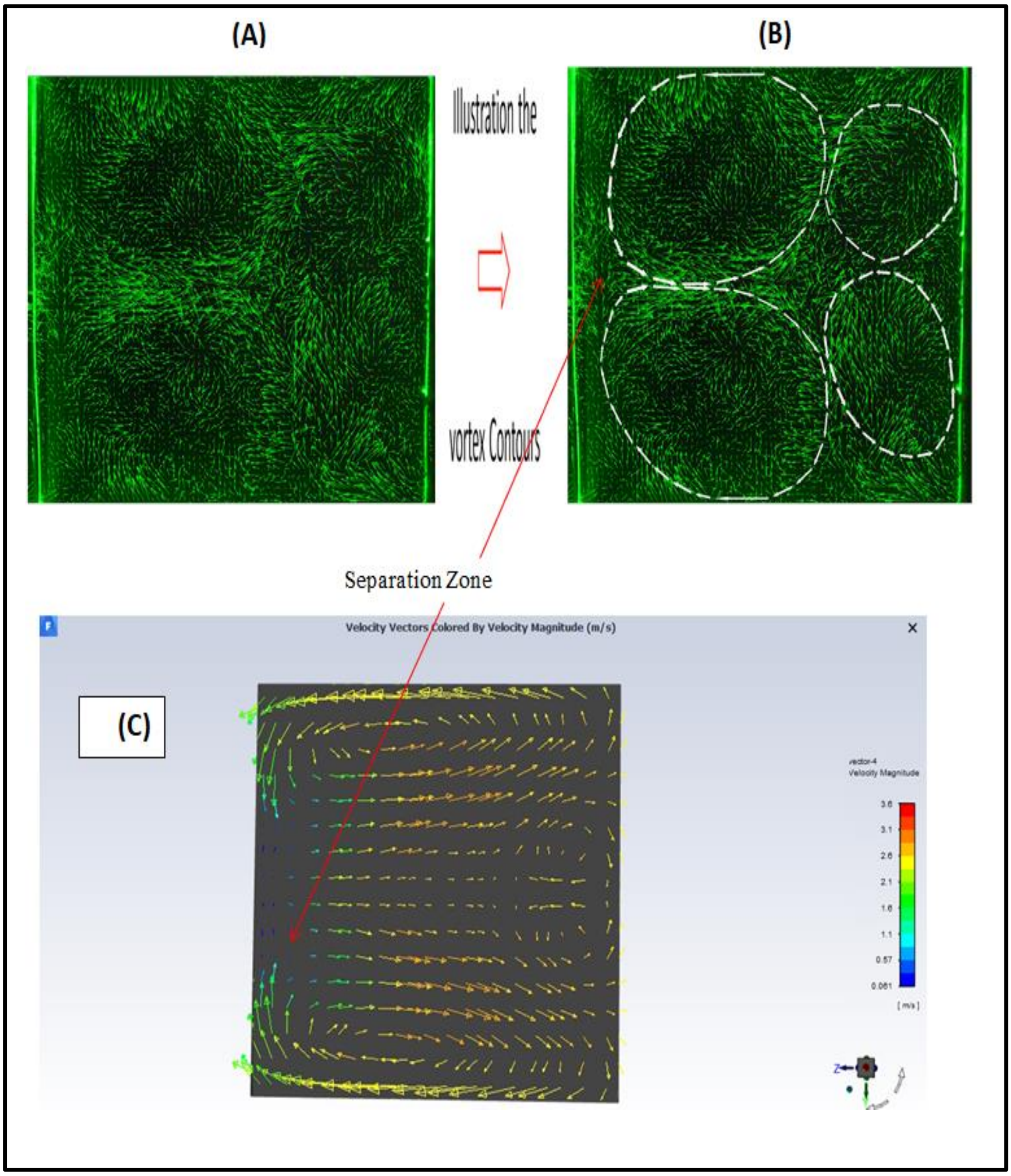


Figure (8): (A and B) Snapshots of the flow structure downstream the 90◦ bent duct, by PIVLAB measurements, with curvature ratio δ=0.5 and test No.(Run 3), (C) flow structure at the same boundary condition obtained by ANSYS Fluent software simulation.

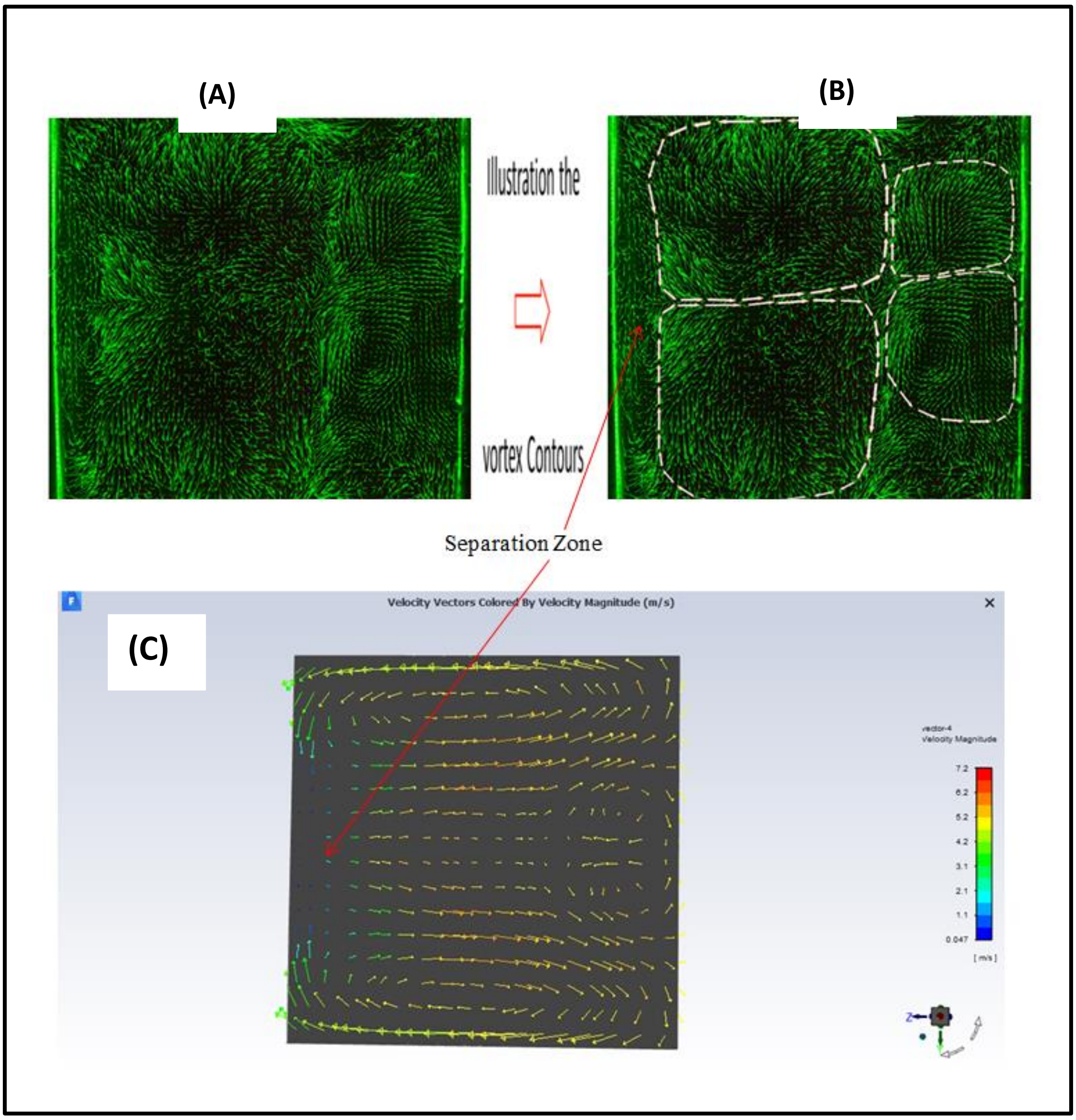


Figure (9): (A and B) Snapshots of the flow structure downstream the 90◦ bent duct, by PIVLAB measurements, with curvature ratio δ=0.5 and test No.(Run 4), (C) Flow structure at the same boundary condition obtained by ANSYS Fluent software simulation.

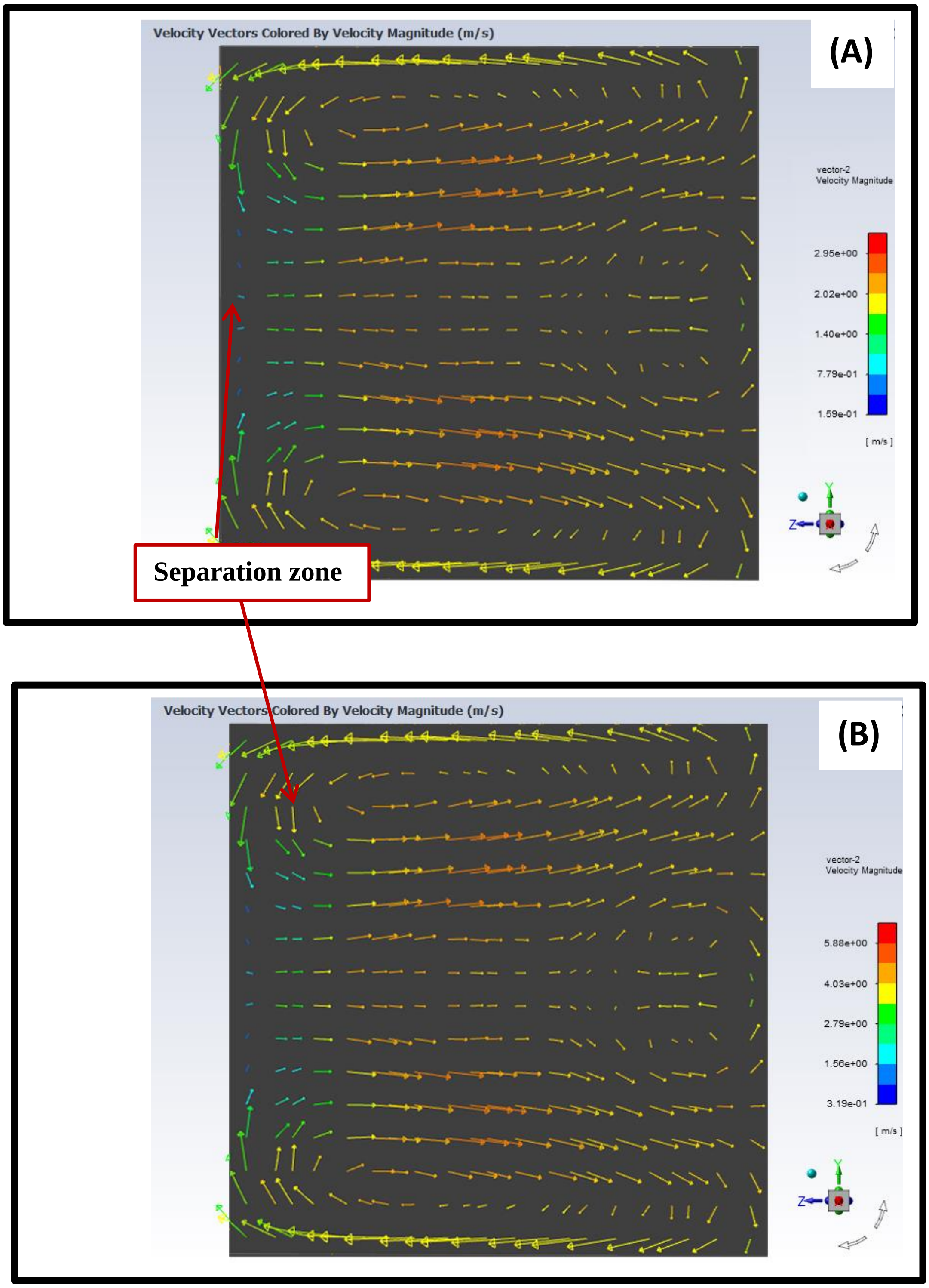


Figure (10): Flow structure downstream the 90o bent duct at curvature ratio δ=0.75 for (A) Run 5, and (B) Run 6, obtained by ANSYS Fluent software simulation.

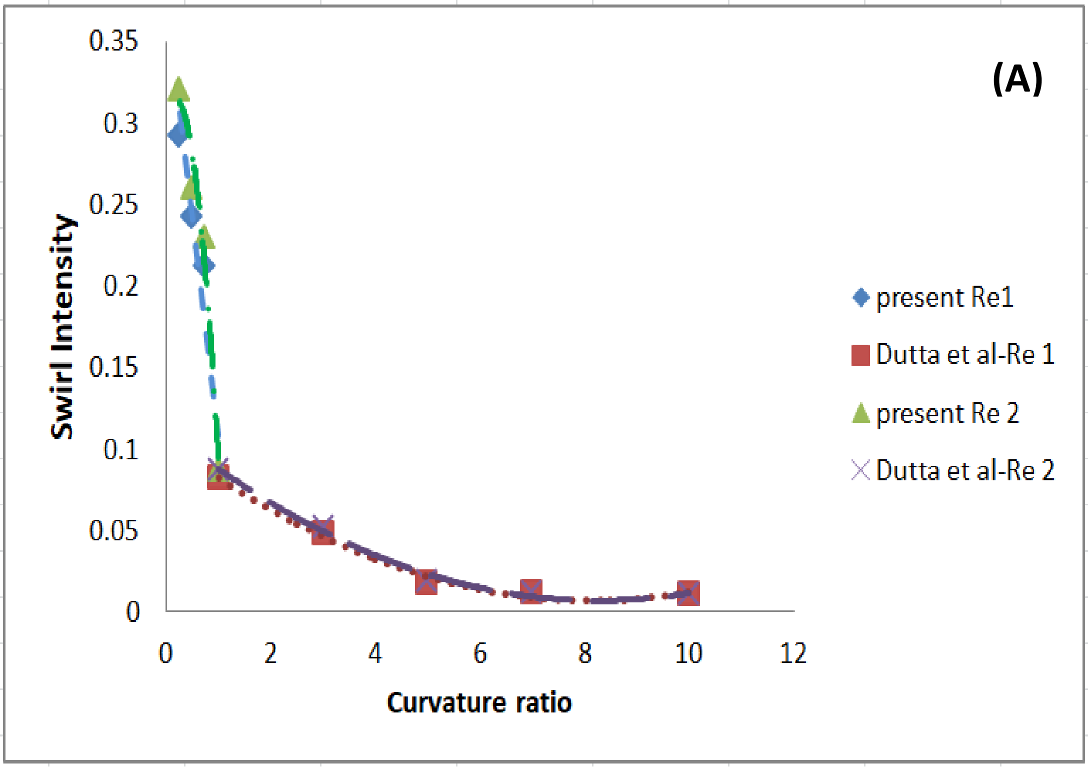


Figure (11): Influence of curvature ratio and Reynolds number on Swirl intensity at bend outlet.

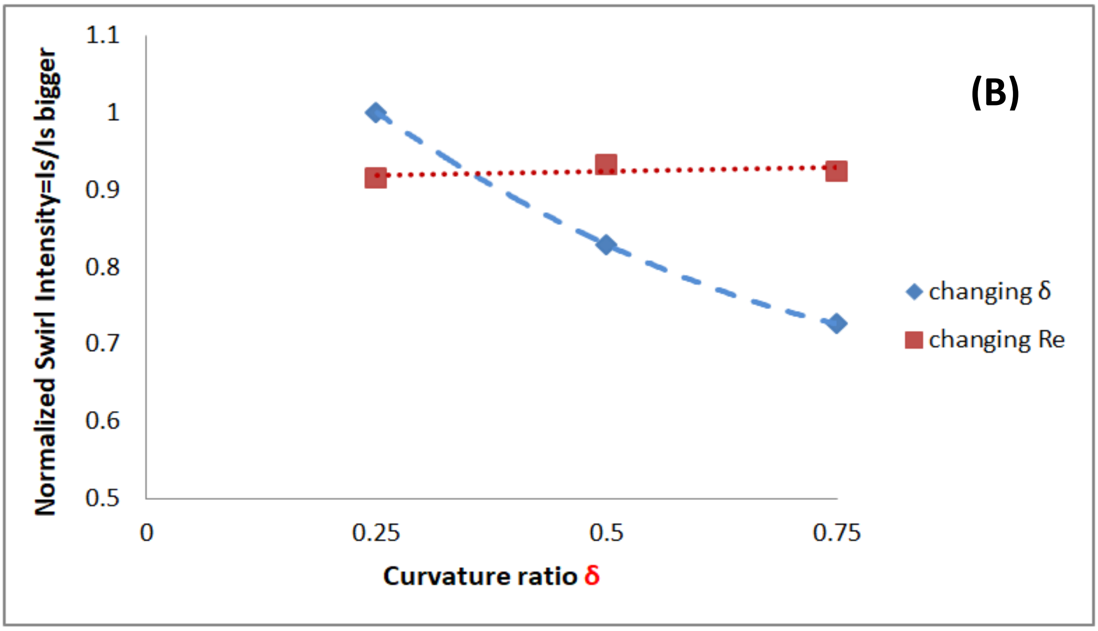


Figure (12): Influence of curvature ratio and Reynolds number on normalized Swirl intensity at bend outlet.

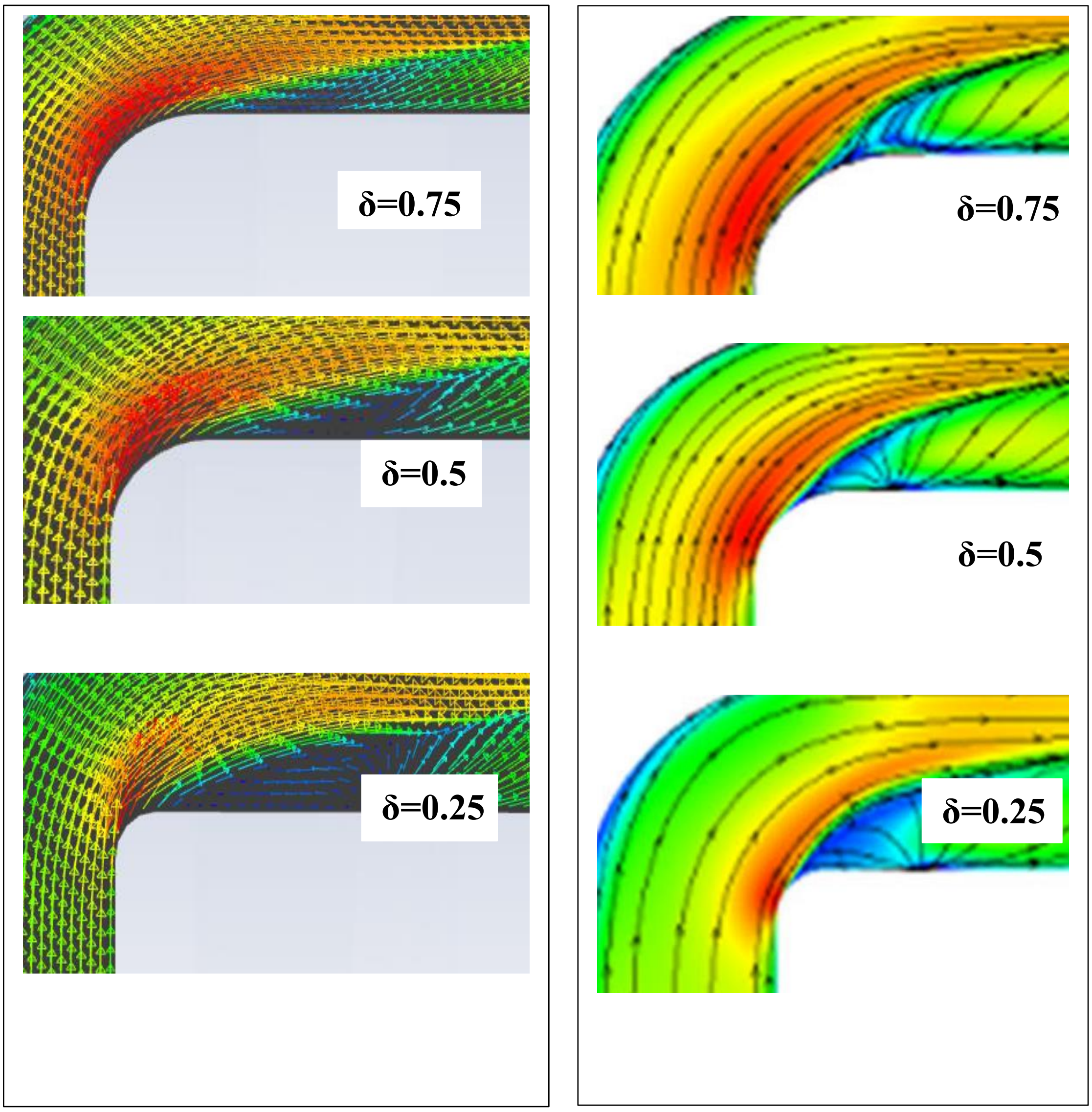


Figure (13): The numerically qualitative comparison of forming the separation zone in the downstream of the bend, for curvature ratios of (δ=0.75, δ=0.5, δ=0.25) from top to bottom; (A) present work; (B) previous work by [4].

Conflicts of Interest

The author declares that they have no conflicts of interest.

7-References

[1] D. Gangopadhyay, N. Kr. Prasad, A.K. Biswas and P.K. Sinha "Flow Investigation inside A Curved Square Duct", Int. Journal of Engineering Research and Application, Vol. 7, pp.17-20,2017.

[2] M. Boutabaa, L. Helin, G. Mompean, and L. Thais, "Numerical study of Dean vortices in developing Newtonian and viscoelastic flows through a curved duct of square cross-section", C. R. Mec. 337, 40 (2009).

[3] P. Dutta and N. Nand, "Effect of Reynolds Number and curvature ratio on single phase turbulent flow in pipe bends", Mechanics and Mechanical Engineering, Vol. 19, No. 1 (2015).

[4] P. Dutta, S. K. Saha and N. Nandi, "Numerical study of curvature effect on turbulent flow in 90˚ pipe bend" Dep. of Aerospace Eng. and Applied Mechanics, ICTACEM-2014/028.

[5] P. Dutta and N. Nandi, "Numerical Study on Turbulent Separation Reattachment Flow in Pipe bends with Different Small Curvature Ratio", J. Inst. Eng. Ser. C, vol. 100, no. 6, pp. 995–1004, 2019.

[6] M. Norouzi and N. Biglari, "An Analytical Solution for Dean Flow in Curved Ducts with Rectangular Cross Section", Physics of Fluids, 25.5 (2013).

[7] Athanasia Kalpakli, "Experimental study of turbulent flows through pipe bend", Technical Reports from Royal Institute of Technology KTH Mechanics SE-100 44 Stockholm, Sweden, 2012.

[8] Yan Wang, Quanlin Dong, and Pengfei Wang, 'Numerical Investigation on Fluid Flow in a 90-Degree Curved Pipe with Large Curvature Ratio', Mathematical Problems in Engineering, 2015 (2015).

[9] A. Ono, N. Kimura, H. Kamide, and A. Tobita, "Influence of elbow curvature on flow structure at elbow outlet under high Reynolds number condition," Nuclear Engineering and Design, vol. 241, no. 11, pp. 4409–4419, 2011.

[10] Kim, J., Yadav, M. and Kim, S., "Characteristics of Secondary Flow Induced by 90˚ Elbow in Turbulent Pipe Flow", Engineering Applications of Computational Fluid Mechanics, 8(2), 229–239, 2014.

[11] Abdul Satar J. Mohammed, Sadiq G. Abed-Alfathel, "The Effect of Curvature Ratio on the Evaporative Cooling of Air Flow Through Bent Duct", Kufa Journal of Engineering, 11.1 (2020).

[12] ASHRAE, "HVAC Systems and Equipment ", Handbook, ASHRAE Inc., Atlanta; 2000.

[13] Thielicke, W and Stamhuis, "PIVLAB – Towards User-friendly, Affordable and Accurate Digital Particle Image Velocimetry in MATLAB". Journal of Open Research Software, E J 2014.

تأثير نسبة الانحناء على بنية الجريان وخلط الموائعي قناة مربعة منحنية بزاوية (90°)

عبد الستار جواد محمد

قسم الهندسة الميكانيكية، الجامعة التكنزلوجية، بغداد-العراق

asj_2006saraf@yahoo.com

صادق جهام عبد الفاضل

قسم الهندسة الميكانيكية، الجامعة التكنزلوجية، بغداد-العراق

saduqalshibly@yahoo.com

الخلاصة

كثير من المشاكل الميكانيكة والكيمياوية تعتمد بالأساس على خصائص الأختلاط لسائل مرشوش مع الغاز الحامل له وهي عملية تتأثر بقوة بمعدلات تبادل الكتلة والحرارة. الجريان الثانوي الناتج عن قوى الطرد المركزي المصاحبة لتغير جوهري في اتجاه الجريان يسبب تكون دوامات متعاكسة الدوران. تهدف الدراسة الى التحقق من أثر نسبة الأنحناء على بنية الجريان وشدة الاضطراب خلال عملية خلط سائل مع غاز لمنظومة حقن ماء مسبقة لمجرى منحني. تستثمر الدراسة تقنية المعالجة الصورية التجريبية PIV بهدف تعقب عملية تكون جريان ثانوي اثناء انتقال خليط ماء–هواء عبر جزء منحني من المجرى. أعتمدت الدراسة ثلاث نسب أنحناء 0.25،0.5 و0.75 لمتوسط سرع 2.5 و 5 m/s لجريان هواء عبر مجرى مربع. تظهر الصور تكون زوج من دوامات دوارة (رباعية–الفصوص) لكل نسب الأنحناء المدروسة مع اندفاع الدوامات القريبة من الجدار الداخلي للمنحني نحو الخارج مع تناقص نسبة الأنحناء كنتيجة لتأثير الطرد المركزي وأنفصال الجريان. تصميم وترتيب مصفوفة حاقنات الماء تقرر أستنادا الى المحاكاة العددية بأستخدام الحزمة البرمجية ANSYS FLUENT 19.R1 مع اعتماد نموذج الأضطراب RNG-k-. النتائج العددية تظهر ان شدة التدويم لها تأثير صغير على عملية الخلط عند تغير عدد رينولدز ولكنها تتأثر بقوة مع تغيير نسبة الأنحناء. المقارنة الظواهرية بين النتائج التجريبية والعددية قدمت تقاربا جيدا للدراسة الحالية، حيث أن الحد الأقصى للانحراف المسجل كان (7,1)%.

الكلمات الداله:– مجرى منحي، نسبة الانحناء، هيكل التدفق، تقنية (PIVLAB)، خلط السوائل.