Experimental and numerical investigations for turbulent airflow characteristics of circular orifice plate

N Abed¹, H F Hassan², Amer Al-damook³, W H Khalil³
¹Mechanical Technical Department, Technical Institute of Anbar, Middle Technical University, Iraq
²Chemical Industries Technical Department, Technical Institute of Anbar, Middle Technical University, Iraq
³The Renewable Energy Research Centre, University of Anbar, Iraq

Abstract. The present work describes the airflow characteristics upstream and downstream of the circular orifice numerically using the standard k-ε model and RSM and experimentally employing hot-wire and Pitot tube devices to measure the flow velocities through a tube. An expectation has to be in mind; the RSM performed well and predicted results in a good level of agreement against the experimental measurements found in preceding studies and the present experimental data as well. In terms of flow physics, velocity and turbulent kinetic energy distribution are presented. Moreover, turbulent viscosity, Reynolds stresses (u², v², w²) are also considered. Furthermore, the contours of velocity, static pressure and turbulent kinetic energy are presented beside the wall shear stress and y+ profiles are illustrated to show the flow physics as well as to compare the results of two different turbulence models considered in this study employing Reynolds of 30000 and the ratio of orifice diameter to the pipe diameter of 0.5.

Keywords: Orifice plate device, hot-wire and Pitot tube devices, RANS models, turbulent airflow characteristics.

Highlights:
1. Turbulent flow characteristics through an orifice plate.
2. Comparison between the standard k-ε model and RSM performances.
3. Experimental investigations using hot-wire and Pitot tube devices.

1. Introduction
The orifice plate device is widely used equipment for measuring the flow characteristics in engineering gas and oil fields. However, the flow measurements in such fields should be recorded accurately and economically. The basic idea of using the orifice plate is depending on the pressure drop before and after the orifice plate owing to the fact of the reduction in the cross-sectional area which is usually represented by the aspect ratio ($\beta$=drifice/ Dpipe).

The flow behaviour becomes more complicated in circumstances some conditions are inserted like an orifice plate, sudden expansion, sudden contraction, incompressible flow and flow viscosity etc. and consequently, the Navier-Stokes closure would not be easy to solve. Meanwhile, the orifice plate cannot only use to manage the flow but further measuring the flow characteristics. On the other hand, some errors in the flow measurements would be resulting in huge cost on the industry/ engineering sectors budget and; therefore, one should pay attention to dealing with such fields [1]. Due to the fact that the flow behaviour through an orifice plate is more complex and it is not easy to capture the flow
physics just beyond the orifice plate owing to the separation and reattachment flow, there are insufficient researchers have taken place in turbulence modelling point of view while a good level of attention has been given to the applications of this type of flow such as industrial applications, medicinal purposes, and agriculture works.

As long as there is a turbulent flow takes place through such applications, it is essential to apply a turbulence model beside the Navier-Stokes equations. One of the important turbulence models is the standard $k$-$\varepsilon$ model which is modelled basically for the simple flow since it would be expected to predict a huge amount of turbulent shear stress while the Reynolds stress model (RSM) is expected to perform better due to its strengths in terms of employ equations more than those employed by the standard $k$-$\varepsilon$ model; nevertheless, the RSM model is computationally expensive and long-time is required to converge [2]. Most of the following CFD simulation studies have been used FLUENT-CFD to investigate the turbulent airflow characteristics.

Experimental data through an orifice plate at Reynolds number of 35000 was reported by Durst and Wang [3]. It was found that the reattachment point had occurred at $x/D=3$, where ($x$) is the axial distance (m) and ($D$) is the pipe diameter (m). The standard $k$-$\varepsilon$ model has been considered in a numerical investigation to find the influence of the Reynolds number ($\rho UD/\mu$) and aspect ratio on the mass transfer rate close to the wall downstream of the orifice plate has been considered by El-Gammal et al. [4]. Results revealed that as increasing the $Re$ number from the increasing Sherwood number (Sherwood number = $h_d d/D$, where $h$ is the local mass transfer coefficient, $d$ is the orifice diameter and $D$ is the pipe diameter) is achieved by 60%.

Shah et al. [5] have studied numerically the flow features through an orifice plate using the standard $k$-$\varepsilon$ model. It was looking for estimating various turbulent flow performances and results had been compared against preceding experimental investigation using air as fluid. Good agreement has been achieved. Authors are recommended to use CFD simulation as a cheap tool and alternative technique insisted of experiments. The stress-blended eddy simulation (SBES) and standard $k$-$\omega$ turbulence models were employed to predict the properties of disturbed pipe turbulent air flows of a segmental orifice plate by Straka et al. [6]. A comparison had been performed between results of both turbulence models against those of the laser Doppler anemometry (LDA) and ultrasonic meter measurements. The findings show that the SBES is superior to other methods of experimental measurements.

Geng et al. [7] used the centric and the eccentric orifice plates to estimate the measurement uncertainty of a liquid mass flow. The comprehensive of two approaches of uncertainty analysis: the Guide Uncertainty Measurement (GUM) and the numerical method of Monte Carlo were considered. The mass flow uncertainty of GUM was larger by 0.04% than that of Monte Carlo simulation. The comparisons between experimental study and different turbulence models (RSM, the standard, RNG and Realizable $k$-$\varepsilon$ models) were investigated by Reis et al. [8] to estimate the burner orifice turbulent gas flow. It was indicated that the results of turbulence models were in good agreement with the measurements in literature. Nevertheless, the most suitable model was the Realizable $k$-$\varepsilon$ model, which its prediction was very similar to those predicted by the RSM and RNG $k$-$\varepsilon$ models. The CFD simulation and experimental test were investigated to consider the effect of the fractal flow conditioner (it is a device which can be utilized before the orifice plate to remove the swirl and create repeatable velocities in the downstream direction) together with an orifice device on the turbulent airflow characteristics by Manshoor et al. [9]. The both flow conditions (standard and non-standard) had been achieved under different mass flow rates and Reynolds number. The findings show that the useful design of the fractal plate has been obtained as porosity is 51.58% and set in a distance of 5 times a pipe diameter upstream of an orifice device. Furthermore, it can dampen out the flow distortion because of the swirling flow and asymmetric velocity profile.

With respect to the variable area of the orifice plate, Singh et al. [10] estimated the differential pressure and the drag force. They found that the differential pressure increases with increasing the blockage of orifice plate due to acting great drag force on the orifice plate. However, within the flow rate of 1-10 kg/s, the differential pressure was not a significant variation. Thus, it was recommended more detail about the flow meter characteristics investigation. Ahmed et al. [11] investigated the
impact of mass flow rate and local flow on flow that accelerated corrosion (FAC) downstream of an orifice plate. The FAC wear rate had been strongly influenced by the geometry of the orifice. The highest value of the wear rate was discovered to be placed within a distance of 5 times the pipe diameter downstream of the orifice. Furthermore, wear rate has increased with reducing the diameter of orifices.

The effect of the perforated plates on the minimizing the combustion of energy, due to reducing the permanent pressure loss, has been investigated by Shaaban [12]. They compared with other differential type flow meters. The numerical data showed that the permanent pressure loss reduced by 51.7% at Re=35000. The effect of different shapes of the fractal orifice on pressure drop losses and the axial velocity were considered by Elsaey et al. [13] to enhance flow measurements techniques. Results revealed that the standard k-ε model which is used in this study is in good occurrence with other measurements. The pressure drop losses of fractal orifice shaped are lower 1.25% than that of the circular orifice plate. Moreover, the axial velocity scales was wider than that of the circular orifice. However, the single and two-phase flows across the slotted orifice plate were numerically and experimentally considered to develop a wet gas meter by Geng et al. [14]. It was pointed out that the metering characteristics of the slotted orifice are superior to the circular orifice. Furthermore, the numerical and experimental data have been in good occurrence. Two types of perforated orifices with circular and rectangular perforations were investigated to estimate the slotted orifice performance by Kumar and Bing [15]. The CFD data indicated that the static pressure recovery and pressure drop of the perforated orifices were enhanced and more sensitive to gas flow rates compared with the standard orifice plate. However, the differential pressure does not seem significantly changeable with the perforation of the orifice.

The main objectives of this study are to experimentally gain familiar with flow behaviour including its separation, recirculation and reattachment as well as the adverse pressure gradient. More details of turbulent airflow characteristics (the turbulent intensity, velocity profiles, volume flow rate, axial velocity, turbulent kinetic energy, wall shear stress, wall y+, Reynolds stresses and pressure contours) in a complex flow are numerically considered by employing two turbulence models; the standard k-ε model with low Reynolds number / wall function treatment and RSM model. Furthermore, a comparison between results obtained by the hot-wire device and those achieved by the Pitot tube with the numerical data.

2. Experimental Method

2.1 Experimental Apparatus and Measurement Devices

The experiment apparatus is a collection of several tools. Basically, an axial fan is connected with a pipe which has been divided into two parts; one in which has an internal diameter of 98 mm and the second part with 102 mm internal diameter connected with each other by a flange, as shown in Figure 1(a). The length of two pipes is 8.976 m which are appropriate to provide fully-developed turbulent flow. This pipe is joined with a plastic pipe, which has a length of 308 mm and is appropriate to provide reattachment turbulent flow, between these two pipes, there is an orifice plate which will cause the separation flow and make the behaviour of flow more complicated due to the sharp change in the cross-section area. There are different scales are used to limit the section that we need to measure the velocity and turbulence intensity, these scales are represented by small plates with scales of (4h, 6.8h, 8.8h, and 10.8h), as shown in Figure 1(b).

A hot-wire probe is used to measure both mean and oscillating velocities in the fluid flows, which is used to measure the significantly changeable velocities at small applications. It has been placed inside the pipe with suitable traverse gear used to increase the distance from wall forward to the pipe centre and vice versa, the hot-wire is joined with two digital voltmeters (one of them for mean voltage while another one is for fluctuating voltage) and with the constant temperature anemometer in order to measure the average and oscillating velocities, the calibration of hot-wire is done by a wind tunnel. Another tool is employed at probable reattached region in order to measure pressure drop which is Pitot tube where the tube with small diameter is placed in the front of the pipe exit where the small tube is placed in the opposite direction with the flow direction, the pressure drop obtained from Pitot
tube will be used in Bernoulli equation in order to achieve velocity which will be compared with this obtained by hot-wire.

(a) (b) (c)

Figure 1. (a) The collection of tools used in the experiment (b) the positions of measuring the velocity and turbulence intensity by hot-wire in the plastic pipe, and (c) computational domain and boundary conditions.

2.2 Experimental Test Procedure

Initially, it is important to calibrate the hot-wire using wind tunnel and the voltage, velocity and pressure can be recorded in order to produce the calibration curve between voltages and velocity. After that the hot-wire can be placed very close to the pipe wall in parallel direction, it would be 1 mm for initial reading, and make sure to place the first scale (plate of 4h=100mm) between the plastic pipe and the gear, and switch on the axial fan, when the case is stable which be obvious from the profile of the boundary layer on the screen take the first reading for voltmeter, see figure 5. Then increase the distance by transverse gear to 2 mm and do the same procedure as above until the measurement would be taken at centreline. After that, another scale (plate of 6.8h=170mm) can be used and take the
measurements at each even number and so on until also reaching the centreline and do the same with other scales.

3. Computational Method

3.1 Physical Domain

Axisymmetric two-dimensional simulations have been performed to reduce the time of the converged solution and the size of the computational model, as shown in Figure 1(c). An orifice diameter (d=0.5D) is nearly 0.5 pipe diameter (D= 0.1m). The domain is considered approximately 1.5D upstream of the orifice plate to achieve the velocity distribution in the fully developed region in all domains. While the downstream of the orifice plate nearly 7D downstream of the orifice plate; therefore, one can make sure that the zero gradients can be applied as a boundary condition. Furthermore, to avoid separation/reattachment region and the negative effect solution around the plate.

3.2 Turbulence Modelling

The flow was numerically solved by the standard k-ε model by employing a low Reynolds number near-wall model and refined the mesh close to the orifice plate. The flow after that is solved with the same mesh by the Reynolds stress model (RSM). Following Launder and Spalding [16], the standard k-ε model was employed firstly in order to gain convergence early since just two differential equations are involved. It can be described as the simplistic turbulence model which can be used with only initial conditions and boundary conditions are required. It has been broadly validated in many types of flow applications and it has only two transport equations; first is for the turbulent kinetic energy (k) and the second equation is for the dissipation energy rate (ε). Both equations are employed to introduce the velocity scale (ν) as well as the length scale (ℓ) in the following procedure, Launder and Spalding [16]:

\[ \nu = k^{1/2} \quad \& \quad \ell = \frac{k^{3/2}}{\varepsilon} \]

The equations for turbulent kinetic energy (k) and the dissipation rate (ε) are given by:

\[
\frac{Dk}{Dt} = \frac{\partial k}{\partial t} + \frac{\partial (\overline{u_i}k)}{\partial x_j} = P_k - \varepsilon + \frac{\partial}{\partial x_j} \left[ (v + \nu) \frac{\partial k}{\partial x_j} \right] \tag{1}
\]

\[
\frac{D\varepsilon}{Dt} = \frac{\partial \varepsilon}{\partial t} + \frac{\partial (\overline{u_i} \varepsilon)}{\partial x_j} = C_{\varepsilon 1} \frac{\varepsilon}{k} P_k - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ (v + \nu) \frac{\partial \varepsilon}{\partial x_j} \right] \tag{2}
\]

Where the production of turbulent kinetic energy (P_k) is provided by:

\[ P_k = -\overline{u_i'u_j} \frac{\partial U_i}{\partial x_j} \tag{3} \]

However, the (P_k) is conjugated with the Boussinesq hypothesis and given by:

\[ P_k = -\overline{u_i'u_j} \frac{\partial U_i}{\partial x_j} = 2\nu_i S_j S_j \tag{4} \]

Whereas the turbulent viscosity (ν_t) and strain rate (S_j) are given respectively by:

\[ \nu_t = C_{nu} \frac{k^2}{\varepsilon} \quad \& \quad S_j = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_i}{\partial x_j} \right) \tag{5} \]
The other parameters are taken as constant values: $C_\mu = 0.09$, $C_{\varepsilon_1} = 1.44$, $C_{\varepsilon_2} = 1.92$, $\sigma_k = 1.3$, $\sigma_\varepsilon = 1$.

For the Reynolds Stress Model (RSM), it considers the complex flow such as the influences of streamline curvature behaviour swirl performance, rapid changes in strain rate and rotation better accuracy than one-equation and two-equation models, while being computationally cheaper compared with Large Eddy Simulations (LES) and Direct Numerical Simulations (DNS). RSM has six different equations are found according to Eiamsa-ard et al. [17], as follows:

$$\frac{\partial \tau_{ij}}{\partial t} + \frac{\partial (u_k \tau_{ij})}{\partial x_k} = -G_{ij} - \Phi_{ij} + D_{ij} + \varepsilon_{ij} \tag{6}$$

Where $(G_{ij})$ represents the local production of the Reynolds stresses and can be expressed by the following expression:

$$G_{ij} = \rho P_{ij} = -\left(\frac{\mu_2}{\sigma_k} \frac{\partial u_i}{\partial x_k} + \frac{\mu_1}{\sigma_k} \frac{\partial u_l}{\partial x_k}\right) \tag{7}$$

The second term in the equation (6) ($\Phi_{ij}$) is the local pressure strain and can be given by the equation below:

$$\Phi_{ij} = -C_1 \frac{\rho e}{k} \left(\frac{u_i}{u_l} - \frac{2}{3} k \delta_{ij}\right) - C_2 \left(\frac{G_{ij}}{2} G_{ij}\right) \tag{8}$$

The last terms in the equation (6) ($D_{ij}$ and $\varepsilon_{ij}$) are the net diffusivity of the Reynolds stress and the local dissipation tensor respectively, which can be formulated as follows:

$$D_{ij} = -\frac{\partial}{\partial x_k} \left(\frac{\mu_k}{\sigma_T} \frac{\partial u_l}{\partial x_k}\right) \tag{9}$$

$$\varepsilon_{ij} = \frac{2}{3} \rho \varepsilon \delta_{ij} \tag{10}$$

The model constants are taken as $C_1 = 2.5$ and $C_2 = 0.55$, Eiamsa-ard et al. [17].

where $\rho$ is density, $\mu_e = \mu + \mu_t$ is the effective viscosity, $\frac{\partial \tau_{ij}}{\partial t}$ is the time rate of change the Reynolds stress at a fixed point, $\frac{\partial (u_k \tau_{ij})}{\partial x_k}$ is the net convection of Reynolds stress by the mean flow to the fixed point, $G_{ij}$ is the local production of Reynolds stress, $\Phi_{ij}$ is the local pressure strain, $D_{ij}$ is the net diffusive transport of Reynolds stress to a fixed point and local dissipation tensor.

3.3 Boundary Conditions

The size of computational domain is reduced due to asymmetry simulations conditions. The periodic flow is used between the inlet and outlet boundaries to ensure the fully developed flow can be achieved on a short domain. A prescribed flow rate and Reynolds number are 0.04216kg/s and 30000, respectively. All remaining walls are applied to be no-slip ($U=0$) and adiabatic ($Q=0$) conditions, see Figure 1(c).

3.4 Solution Procedure and Convergence Criteria

A commercial finite volume method (FVM) ANSYS FLUENT [2] is used to solve the time-averaged Navier-Stokes equations, the turbulent kinetic energy equation, and the turbulent energy dissipation rate equation in addition to the turbulence model equations, the standard $k-\varepsilon$ model or for the RSM. The SIMPLE method is used for the continuity ($\frac{\partial (\rho u_i)}{\partial x_i} = 0$). Computation was carried out to start first by solving the continuity, momentum, $k$ and $\varepsilon$ equations or RSM equations to determine the air flow characteristics and the flow field in the computational domain. The procedure continues until the sum of the residuals of continuity and momentum equations in each cell is less than 10$^{-5}$.
3.5 Post processing

Air flow characteristics through an orifice plate are important parameters, in which the values are calculated from ANSYS FLUENT post-processing options. These parameters such as mean velocity, dimensionless radial ratio \((r/R)\), turbulent kinetic energy \((k)\) and dimensionless average stream-wise velocity ratio \((U/U_{\text{max}})\), \(y^+\), wall shear stress \((\tau_w)\) and turbulent intensity \((I)\) along the pipe at a number of stream-wise locations are considered in this work, where \(R\) is the main pipe radius and \(U_{\text{max}}\) is the maximum velocity.

3.6 Grid Independent Tests (GIT)

Quadrilateral cells are used for a number of various grid distributions to ensure the achievement of grid verification and the reliability of mesh density. Thus, the CFD simulations are verified for the orifice plate by comparing the turbulent kinetic energy distribution. In this case, the grid generation should be refined near the solid faces of the orifice plate to predict the accurately characteristics of turbulent flow. The coarse and fine meshes have been compared with the experimental results, as shown in Figure 2. The findings show that the fine mesh is closer to the experimental results than that of the coarse mesh. Hence, in this study, the fine mesh is selected as a standard for grid independent tests for all the predicted simulations data of the orifice plate.

![Figure 2](image_url)

**Figure 2.** comparison grid independent tests of turbulent kinetic energy with the experimental results.

4. Result and Discussion

4.1 Normalised velocity profiles at different locations

In spite of the fact that the Hot-wire anemometer has extremely high frequency-response and fine spatial resolution compared to other measurement methods, it is unable to detect the flow reversal in recirculation region, since the hot-wire anemometer is insensitive to the flow direction in this region, and as the flow has passed the orifice plate, the flow becomes more strong and there is a backflow of the air. Therefore, as hot-wire anemometer is used, it is necessary to deduce that the flow direction by means of the pressure drop or by correcting the results depending on the volumetric flow rate. In order to correct the results in this work, it is looking for the result of volume flow rate which should be the same in all taken sections. That is done by making the area under the curve of mean velocity multiplied by radius versus radius and; as a result, after some interpreting, it is obtained approximately the same flow rate, see Figure 3.
The reference point was a section (10.8h) because it is ensured that the flow at this location has definitely reattached and no backflow can occur; consequently, the hot-wire anemometer is able to measure probably in this region. Another solution one might use another type of hot-wire anemometer which has an ability to measure the mean and fluctuating velocities in recirculation flow.

The extra mechanism was implemented for both fields; momentum and energy due to the effect of turbulent eddies which are rapid fluctuations located in the swirling regions in the flow domain. Consequently, significant oscillations have resulted in various parameters such as temperature, velocity and also density. As can be shown from Figure 4 different places upstream and downstream of the orifice plate the velocity distributions. The parameter (h) represents the height of the orifice plate which is 25 mm whereas (y) means the vertical distance which represents zero at the bottom wall and equals pipe diameter (50 mm) at the top wall.

In all cases, the profiles seem to very flutter and a sharp reduction is noticed near-wall region. Indeed that the viscous sub-layer is very thin while it possess a very important role due to the large velocity gradient in the flow feature. Therefore, the velocity is very small (close to zero) near the wall whereas it is maximum at the tube center, [18]. As can be shown in Figure 4(a), 4(b) and 4(c), the flow behaviour very close the wall has recorded negative values for velocity and these negative values reduced gradually in the downstream direction. This might be due to the backflow of the air since the sudden increase in the static pressure reached the largest value in the region just before and after the orifice plate providing the sudden pressure drop. This drop would be continuing until achieving the point that has the largest velocity and smallest pressure. However, a gradual rise has been noticed in the pressure beyond this point.

On the turbulence modelling side, the results predicted by the RSM are more close to the measurements than those predicted by the k-ε model an especially at the pipe centre. Obviously, that could be happened behind some reasons afforded by RSM includes; both diffusion and convection parts are involved in the Reynolds stress equation (6) and this equation has also body force and production parts and; therefore, the rotation influences can be easily responded. Moreover, the Reynolds stresses predicted in this model are not equal which describe the anisotropic flow and resulted in more realistic behaviour. Besides that, the exacted treat is given to the production term.

The predictions a far from the tube center until the pipe wall are not in a very good level of agreement with the measurements. That could be due to the computation of stress tensor (Rij) in RSM model which is computed locally within the domain cell. On the contrary, the difference between the
measurements and predictions of the standard k-ε mode and RSM is very obvious. In the zone of (100 mm), results of the k-ε model near the wall region are near the measurements as well as at the tube center region; while large values are predicted in between. Nevertheless, the k-ε model was not able to capture the flow characteristics probably in the recirculation flow region as shown in Figures 4(c) and 4(d).

That might be because of the fact that the k-ε model has an assumption that there is a direct relation between the stress tensor and the average strain rate tensor depending on the Boussinesq hypothesis. This assumption can be easily implemented in a simple flow not in such complex flow due to the large pressure drop, sudden contraction and after that faces the sudden expansion. Therefore, this would have a barrier in terms of computing the features of the strongly swirling stream and finally failed to capture the flow characteristics in the zone of recirculation flow. The very crucial feature one can notice from the turbulence models, they performed better than the experimentation in predicted physics point of view particularly RSM since the hot-wire failed to accurately measure direction and magnitude of the flow velocity rather than just the velocity magnitude.

**Figure 4.** comparison of velocity profiles between current experimental and simulation results at different downstream locations; (a) 100 mm, (b) 170 mm, (c) 220 mm, and (d) 270mm.
As shown in Figure 5, there is a difference between calibration equations only at the beginning and it is very clear the quadratic equation is closer to the experimental measurements than that of king’s law; thus, it is selected for calibration purpose. The king’s law can be given by:

\[ E^2 = A + BU^n \]  

(11)

The parameters (A, B and n) are constant depend on the number of points and flow velocity range, Bruun et al. [22].

It is very important to calibrate the hot wire because of several reasons including; the hotwire behaves non-linearity that means the relation between voltage and velocity is non-linear; therefore, the hotwire needs to be calibrated. Moreover, Because of the changing of fluid properties with time, the standard calibration is no longer appropriate for the efficient work. Furthermore, the hotwire relies completely on the difference between the hotwire and ambient temperatures; unfortunately, this perhaps is changed. It is also important in order to avoid the error due to the effect of radiation heat transfer between the pipe metal and hot-wire close to the wall, that might bring a little bit of confusion because of decreasing in temperature is due to radiation heat transfer not because of the convection fluid flow leading to provide high voltages close to the wall.

4.2 Wall shear stress obtained from the simulation investigation:

It has been quite apprehended that the molecular motions and diffusivities are smaller than their turbulent analogues in the turbulent boundary layer. Because of the fact that the no-slip conditions near the wall, the turbulent motion intensity is lost and significantly reduced at the wall (close to zero). Therefore, the velocity distributions have faced a slight change in the centre flow region whereas, at the wall zone, a highly steep change is noticed which in turn led to very high-velocity gradient at the pipe wall [18].

As can be seen from Figure 6(a), the minimum wall shear stress values represent the separation locations of recirculation regions in which a part of the boundary layer has moved inversely to the flow near-wall region. Two different zero values are predicted, one in which located upstream whereas the second value has occurred just downstream with tiny recirculation regions, see precisely Figure 6(b).
Figure 6. (a) the wall shear stress along the pipe wall using the RSM and k-ε models with the fine mesh (b) the path-lines coloured by the velocity shows the recirculation zones.

Meanwhile, the larger recirculation region has placed downstream the orifice plate. However, the separation region can be easily killed and allow to flow to reattach due to the reason of turbulent flow possess momentum and energy more than laminar flow, since the reattachment has occurred at the point of approximately (0.2 or 8h) while in the present work the reattachment point noted to occur in section after the scale of 8.8 step of height (h) downstream of the orifice plate which expects to be at 9h.

However, in the literature, it was implicitly mentioned that the reattachment point in such flow seems to locate at 7h while here it is found something different a little bit. That is due to some reasons including; the hot-wire is not an appropriate tool one can use it in such flow because of inaccurate results provided. Moreover, it is well known that such flow in pipe depends on several parameters such as Reynolds number, the ratio of orifice diameter to pipe diameter and ratio of pipe roughness to pipe diameter etc., therefore, several probable reasons can contribute to producing such difference between present work and the literature. Firstly, the Reynolds number in experimental work is 39000 while in simulation it is just 30000, therefore, it is expected to obtain reattachment point after this obtained by simulation due to high velocity involved. Secondly, the pipe in the experiment consists of two different materials, iron and plastic which have definitely different roughness.

The previous investigations showed that in the expansion pipe diameter (like an orifice plate) that will lead to generating the heat transfer coefficient on downstream location several times larger than that produced fully developed a turbulent flow for the same Reynolds number. This is due to heat transfer augmentation which might be attributed in order to increase the turbulence kinetic energy level of the stream. The very important characteristic is that the sudden expansion in the pipe diameter
can easily produce large shearing rates in the zone which has been removed from the immediate vicinity of the pipe wall. Because of the high level of shearing rate, the turbulence energy generation rates would be definitely larger, whereas the dissipation rates would tend to be very small because of the large scale motions. Therefore, the turbulence level can increase quickly greater than this found in the usual cases. These raise energies would cause increases the turbulent diffusion coefficients in the main (like turbulent) zone of the flow and in the same time it would reduce the thickness of the skin near-wall region and; consequently, the heat transfer would pass in a huge amount by the molecular diffusion, [19]. One can deduct from the above argument that as the skin friction coefficient becomes zero at reattachment point. Consequently, the heat transfer coefficient would be increased gradually downstream and reached the largest value at the point when the flow has reattached.

The wall shear stress distribution is presented in Figure 6(a) employing two turbulence models; the standard k-ε model as well as RSM. It is very clear that the results predicted by the k-ε model are higher than those predicted by the RSM, in which a significant level has been predicted by the k-ε model just the near-wall region and tended to delay the flow separation (major deficiency). This is again due to the presence of adverse pressure gradients as well as recirculation zones which influenced the k-ε model and also because the equation of dissipation involved in this model which provides reasonably unacceptable large length scales near the wall under the diffusion-controlled circumstances.

The flow becomes fully-developed when the axial velocity is no longer change with x-direction. Simply, as it is considered that we are looking at the entrance region of the pipe and we need to compute the fully-developed flow length in turbulent flow, the fully-developed flow length will depend strongly on two parameters which are Reynolds number and pipe diameter in this relation, [20]:

\[ \frac{L_e}{D} \approx 4.4 \frac{\text{Re}}{D}^{1/6} \Rightarrow L_e = 4.4(39000)^{1/6} D \approx 26D \]

In addition to that, in order to reach the reattachment point in such flow, the number of height step is 8h which equal to 4D; therefore, the flow reached the fully-developed case after the orifice plate needs 30D.

4.3 Axial velocity- centreline

When the flow passes the orifice plate, the half of the velocity head is lost since the flow is unable to make sharp 90° turns easily, in particular in the large velocity region; therefore, the flow has separated at the corners and the flow reduced into the region where the vena contracta has formed at the centerline. In this region the velocity reached to a maximum value (with minimum pressure) due to the reduction in the effective flow area and beyond this point, when the flow fills the entire cross section of the pipe, the velocity has decreased and at the same time the velocity head is converted into the pressure head according to Bernoulli’s equation. As shown in Figure 7, the k-ε model predicted the peak mean velocity at (x/h=1.5) and very clear this is far from the experimental measurement whereas the prediction by the RSM is in better level of agreement with the measurements than predictions of the k-ε model.
Figure 7. Comparison between turbulence models and experimental data of the axial velocity along the pipe centreline.

4.4 Turbulent Intensity:
The orifice passage is a typical geometry in which the generation of turbulent energy takes place due to the sudden change of geometrical conditions causing the instability and ultimately culminates in turbulence, as can be seen from Figure 8(a), the turbulent kinetic energy has increased considerably reaching the maximum value just behind the orifice plate at van contact point due to the maximum velocity and very small diameter in this point.

As can be seen in Figure 8(a), the $k-\varepsilon$ model has predicted two peaks of turbulent kinetic energy; first in which occurred due to the high level of turbulent mixing immediately after the orifice plate and then decreased because of the diffusion and then increased to the second peak after the reattachment point (at $x/h = 8$). On the contrary, the RSM has predicted the peak turbulent kinetic energy after the $k-\varepsilon$ model which lies at approximately $x/h=13$ which is the same of the experimental result but still smaller than the experimental peak value since the RSM model gives a long recirculation region than the $k-\varepsilon$ model. The reason for the turbulent kinetic energy decaying after reaching its peak is that the large-scale motions are broken up to small enough scale motion for the dissipation rate to be balanced the diffusion flux.

As shown in Figure 8(b) and (c), still there is a large difference between the results of the fine mesh and measurements even with applying the low Reynolds number near-wall model basically because of the fact that this model has a large level of diffusion (involved in momentum) and large turbulent mixing included. Indeed, the larger the turbulent kinetic energy, the more amount of momentum diffusion would be generated that in turn led to reducing the recirculation lengths (another weakness in this model). In addition to that, the $k-\varepsilon$ model is influenced by two viscous stresses owing to two transport equations of $(k)$ and $(\varepsilon)$, [21].
Figure 8. (a) Comparison between turbulence models and experimental data of the turbulence kinetic energy along the pipe centreline (b) the turbulent kinetic energy predicted by k-ε model and (c) the turbulent kinetic energy predicted by RSM model.

Effectively, what happens in the k-ε model is as it is well known that the production term \( P_k \) can be given as, \([16]\):

\[
P_k = -u_i u_j \frac{\partial U}{\partial x_i} = -u_i^2 \frac{\partial U}{\partial x} - v \frac{\partial V}{\partial y} - w \frac{\partial W}{\partial z}
\]  

(12)

Where \( U, V \) and \( W \) are the mean flow velocities in \( x, y \) and \( z \) directions respectively.

And from continuity:

\[
\frac{\partial U}{\partial x} = -2 \frac{\partial V}{\partial y} = -2 \frac{\partial W}{\partial z}
\]  

(13)
So, substitute equation (13) in equation (12) to give:

\[ P_k = -u^2 \frac{\partial U}{\partial x} + \frac{1}{2} v^2 \frac{\partial U}{\partial x} + \frac{1}{2} w^2 \frac{\partial U}{\partial x} \]  

(14)

From the k-ε model [16]:

\[ u^2 = \frac{2}{3} k - 2 \nu_t \frac{\partial U}{\partial x}, \quad v^2 = \frac{2}{3} k - 2 \nu_t \frac{\partial V}{\partial y}, \quad w^2 = \frac{2}{3} k - 2 \nu_t \frac{\partial W}{\partial z} \]  

(15)

So, substitute equation (13) in equation (15) to give:

\[ u^2 = \frac{2}{3} k - 2 \nu_t \frac{\partial U}{\partial x}, \quad v^2 = \frac{2}{3} k + \nu_t \frac{\partial U}{\partial x}, \quad w^2 = \frac{2}{3} k + \nu_t \frac{\partial U}{\partial x} \]  

(16)

Finally, substitute equation (16) in equation (14) to give the production term:

\[ P_k = -\left( \frac{2}{3} k - 2 \nu_t \frac{\partial U}{\partial x} \right) \frac{\partial U}{\partial x} + \frac{1}{2} \left( \frac{2}{3} k + \nu_t \frac{\partial U}{\partial x} \right) \frac{\partial U}{\partial x} + \frac{1}{2} \left( \frac{2}{3} k + \nu_t \frac{\partial U}{\partial x} \right) \frac{\partial U}{\partial x} \]  

\[ P_k = 3 \nu_t \frac{\partial U}{\partial x} \frac{\partial U}{\partial x} \]  

(17)

On the contrary, results predicted by the RSM are in more agreement with measurements than results of the k-ε model. To be more precise, the RSM's equations are different from those of the k-ε model in which extra equations are present in the RSM leading to capture more realistic results than those captured by the k-ε model.

4.5 The wall (y plus) obtained from the simulation work

Figure 9(a) shows the y-plus profiles predicted by the two turbulence models (RSM and k-ε models). The y-plus describes the friction velocity (=wall shear stress/density), the wall shear stress is highly captured; consequently, the larger value has been predicted by the k-ε model. The y+ value should be near to one as any turbulence model would be employed with wall treatment so as to make sure that the impact of the sub-layer zone has been predicted. However, the y+ value predicted by the k-ε model is still larger than that predicted by the RSM where the larger y+ was 1.2 by the k-ε model while less than 1 (approximately 0.8) captured by the RSM.

Some extra work has been achieved to judge on the most successful turbulence model. Figure 9(b) and (c) present the static pressure contour using both models. It is clear that there is a sudden increase in the pressure reached the largest value just before the orifice plate and after the orifice plate the pressure has suddenly dropped. It is obvious that the gradual decrease in the static pressure predicted by the RSM is better than that captured by the k-ε model.
4.6 The velocity contours

As can be shown from Figure 10(a) and (b), the velocity contours using both models with employing the same fine mesh are presented. The RSM results as shown are more precise in terms of capturing the flow physics in all regions whereas the k-ε model was not able to do so. The RSM performed well and presented good behaviour along the flow pipe started from the pipe inlet until the flow has reached the orifice plate. Additionally, the separation point has captured with a large recirculation zone in the direction of downstream the orifice plate as well as the recirculation length which is longer than results captured by the standard k-ε model. According to the discussion above regarding of many characteristics are achieved by the RSM. However, the k-ε model was unable to obtain it; the most successful turbulence model in separation/reattachment, recirculation flows is Reynolds stress model (RSM).
Figure 10. Velocity contours predicted using fine mesh by (a) RSM and (b) k-ε (c) the comparison between the results of the Pitot tube with hot-wire at the centreline.

4.7 The comparison between the results of the Pitot tube and hot-wire at the centerline

In spite of the fact that the Pitot tube has many advantages such as providing accuracy consistently measurements across different environmental conditions, it can be easily used in tight space since its size is small, it has no moving part, it is also a physically strong tool over other fluid mensuration tools and it is cheaper than the orifice plate. This type of devices has some disadvantages including; as the flow velocity is very small, the difference in pressures is very small, it is very difficult to measure the flow accurately. However, the Pitot tube device as shown in Figure 10(c) is highly insensitive to the flow characteristics in a location of just behind the orifice plate in which the velocity value has reached the largest value while the smallest pressure value has recorded (negative) and; therefore, this device was not able to measure the flow accurately in this region while a far from the effects of orifice plate where the flow velocity tends to decrease gradually due to increasing the flow area, the Pitot tube
tends to mimic the hot-wire and becomes close to it as long as they become far away from the orifice plate.

4.8 The comparison the results of the k-ε and RSM models with measurements

The profile of centerline axial velocity predicted with the LRN (k-ε) and WF RSM models are compared with the measured data in Figure 11(a). The measured data was taken from the region (x/d=1) to the region (x/d=3). A closer examination reveals that the experimental results are in good agreement with those of (Durst and Wang [3]) for the distance compared. The results of the two models used in numerical investigations coincide with each other in the upstream region before the orifice. The use of the WF RSM leads to slight improvement for this flow as can be seen from x/D = 0 to x/D =6 this is due to the ability of the model to predict the flow features influenced by mean of rotational effect (the near walls regions have higher gradients in all variables and the using of wall function can capture the changes in quantities). The LRN (k-ε) fails to predict the flow field for the region (x/d =1 to x/d=4) and its performance is unsatisfactory because it cannot captures the streamline curvature flow and the recirculation zone.

Another interested area is the turbulent kinetic energy distribution captured by the two turbulence models along the flow stream. The average flow is affected by the turbulence, and so a non-real presentation of the turbulence quantities will lead to errors in the mean flow. Figure 11(a) and (b) present the turbulent kinetic energy profiles using low Reynolds number model with both models and the significant increase has been presented through the orifice plate. As the flow across the orifice, the kinetic energy starts to decrease for both models. The major difference between calculations with the two models used is most clear in the downstream orifice region. The decreasing in the LRN k-ε model is greater than that of LRN RSM model. This is because the LRN RSM model accounts for each Reynolds stress unlike the LRN k-ε model, which assumes them to be isotropic (there is considerable under-estimation in the wall normal stress which in turn led to provide poor predicted valued of the turbulence kinetic energy distributions employing the LRN k-ε model). The same interpretation between the two models may be conducted for the wall function. After (0.4 m) the non-dimensional turbulent kinetic energy coincides with both models used in calculations. These agreed with findings of Eiamsa-ard et al. [17].
Figure 11. The comparison between (a) numerical predictions of normalized velocity profile (b) RSM predictions of normalized velocity profile with experimental measurements of the present study and those of Durst 1989.

4.9 Reynolds Stresses
As can be seen from Figure 12(a) and (b), the Reynolds stresses distributions are presented where \( uu \), \( vv \) and \( ww \) are taken into account. It is clear that the maximum contribution to the Reynolds stresses downstream of the orifice plate has been assessed by the square velocity in the \( x \)-direction, i.e. \( uu \), while both other velocities (v and w) provided exactly the same value along the non-dimensional distance. This is due to the fact that the velocity of \( U \) in the \( x \)-direction is much higher than \( V \) and \( W \) and the largest fluctuation has been recorded at the \( x/h=12.5 \).
Figure 12. (a) The comparison between the standard $k$-$\varepsilon$ and RSM predictions of non-dimensional turbulent kinetic energy using Low Reynolds number (LRN) and wall function (WF) models. (b) The predicted Reynolds stresses through the orifice plate using LRN RSM model.

5. Conclusion
The updated numerical and experimental investigations are carried out to examine the effect of such complex flow in the orifice plate using the standard $k$-$\varepsilon$ model and Reynolds stress model (RSM) in terms of CFD simulation while using both hot-wire and Pitot tube devices in terms of experimental measurements. The selection of the turbulence model is very important and has a key role in terms of evaluating the relation between velocity profiles and the turbulent kinetic energy distributions downstream of the orifice device. The $k$-$\varepsilon$ model, as was expected, is not able to predict the accurate
flow physics that can be caused by the anisotropic normal stresses whereas it has been precisely predicted by the RSM model. Nevertheless, the RSM is not quite employed in the industry due to the fact of that this model is computationally expensive and more time is required in order to be in convergence. Therefore, the best model can be selected should be simple, accurate and computationally cheaper.

In terms of the experimental devices, the hot-wire is good tool for measuring flow physics without recirculation issues because the hot-wire can measure the flow just perpendicular on it while with recirculation flow, it is necessary to correct the result or replace the current hot-wire by another one has an ability to measure the velocities where backflow occur. However, the Pitot tube cannot be used in regions have very small pressure because that will lead to producing the wrong values for flow velocity. Moreover, it is suspected that for complex flow in the region of swirl flow and recirculation zones (the gradients of all variables is great) and there is a great anisotropy in the flow field. The RSM model will become superior to the \( k-\varepsilon \) model. This is because that anisotropy can be captured by the RSM model.

**Nomenclature**

**Latin symbols**

| Symbol | Description | Unit |
|--------|-------------|------|
| \( D \) | Pipe diameter | m |
| \( d \) | Orifice diameter | m |
| \( h \) | Orifice height | m |
| \( h_m \) | Local mass transfer coefficient | m/s |
| \( u \) | Fluctuating velocity in x-direction | m/s |
| \( v \) | Fluctuating velocity in y-direction | m/s |
| \( w \) | Fluctuating velocity in z-direction | m/s |
| \( U \) | Mean velocity in x-direction | m/s |
| \( V \) | Mean velocity in y-direction | m/s |
| \( W \) | Mean velocity in z-direction | m/s |
| \( k \) | Turbulent kinetic energy | m^2/s^2 |
| \( P_k \) | Production of \( k \) | m^2/s^3 |
| \( S_{ij} \) | Strain rate | 1/s |
| \( G_{ij} \) | Production of Reynolds stresses | Pa/s |
| \( D_{ij} \) | Net diffusivity | Pa/s |
| \( Q \) | Thermal power | W |
| \( I \) | Turbulence intensity | |
| \( U_{max} \) | Maximum velocity | m/s |
| \( R_{ij} \) | Reynolds stress tensor | |
| \( L_e \) | Fully-developed flow length | m |

**Greek symbols**

| Symbol | Description | Unit |
|--------|-------------|------|
| \( \beta \) | Aspect ratio of orifice diameter to pipe diameter | |
| \( \varepsilon \) | Dissipation energy rate | m^2/s^3 |
| \( \rho \) | Fluid density | kg/m^3 |
| \( \mu \) | Fluid dynamic viscosity | Pa.s |
| \( \nu_t \) | Turbulent viscosity | m^2/s |
| \( \phi_{ij} \) | Local pressure strain | Pa/s |
| \( \tau_w \) | Wall shear stress | Pa |

**Abbreviations**

| Abbreviation | Description |
|--------------|-------------|
| RSM | Reynolds stress model |
| CFD | Computational fluid dynamics |
| Re | Reynolds number |
| SBES | Stress-blended eddy simulation |
| LDA | Laser Doppler anemometry |
| GUM | Guide Uncertainty Measurement |
| FAC | Flow accelerated corrosion |
| DNS | Direct Numerical Simulations |
| LES | Large Eddy Simulations |
| FVM | Finite volume method |
Acknowledgment
The authors would like to thank the University of Manchester for the technical support and many thanks are sent to Dr. Amer Al-Damooq who helped the authors to present this work.

Reference
[1] Siba M. A., Mahood W. M., Nuawi Z. M., Rasani R., Nassir M. H., 2015. Modelling and applications of 3d flow in orifice plate at low turbulent Reynolds numbers. *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS*. **15**(4), pp: 19-25.
[2] ANSYS FLUENT User's Guide, 2011.
[3] Durst F., Wang A. B. 1989. Experimental and numerical investigation of the axisymmetric, turbulent pipe flow over a wall-mounted thin obstacle. In *Seventh Symposium on Turbulent Shear Flows*. Stanford University, **1**, pp: 10.4.1-10.4.6.
[4] El-Gammal M, Ahmed WH, Ching CY. 2012. Investigation of wall mass transfer characteristics downstream of an orifice. *Nuclear Engineering and Design*. **242**, pp: 353-360.
[5] Shah MS, Joshi JB, Kalsi AS, Prasad CS, Shukla DS. 2012. Analysis of flow through an orifice meter: CFD simulation. *Chemical engineering science*. **71**, pp: 300-309.
[6] Straka M, Fiebach A, Eichler T, Kaglin C. 2018. Hybrid simulation of a segmental orifice plate. *Flow Measurement and Instrumentation*. **60**, pp: 124-133.
[7] Geng Y, Zheng J, Shi T. 2006. Study on the metering characteristics of a slotted orifice for wet gas flow. *Flow Measurement and Instrumentation*. **17**(2), pp: 123-128.
[8] Reis LC, Carvalho Jr JA, Nascimento MA, Rodrigues LO, Dias FL, Sobrinho PM. 2014. Numerical modeling of flow through an industrial burner orifice. *Applied Thermal Engineering*. **67**(1-2), pp: 201-213.
[9] Manshoor BB, Nicolleau FC, Beck SB. 2011. The fractal flow conditioner for orifice plate flow meters. *Flow Measurement and Instrumentation*. **22**(3), pp: 208-214.
[10] Singh SN, Gandhi BK, Seshadri V, Chauhan VS. 2004. Design of a bluff body for development of variable area orifice-meter. *Flow measurement and Instrumentation*. **15**(2), pp: 97-103.
[11] Ahmed WH, Bello MM, El Nakla M, Al Sarkhi A. 2012. Flow and mass transfer downstream of an orifice under flow accelerated corrosion conditions. *Nuclear Engineering and Design*. **252**, pp: 52-67.
[12] Shaaban S. 2015. On the performance of perforated plate with optimized hole geometry. *Flow Measurement and Instrumentation*. **46**, pp: 44-50.
[13] Elsaey A, Aly AA, Fouad M. 2014. CFD simulation of fractal-shaped orifices for flow measurement improvement. *Flow Measurement and Instrumentation*. **36**, pp: 14-23.
[14] Geng Y, Zheng J, Shi T. 2006. Study on the metering characteristics of a slotted orifice for wet gas flow. *Flow Measurement and Instrumentation*. **17**(2), pp: 123-128.
[15] Kumar P, Bing MW. 2011. “A CFD study of low-pressure wet gas metering using slotted orifice meters. *Flow Measurement and Instrumentation*. **22**(1), pp: 33-42.
[16] B. E. Launder and D. B. Spalding. 1972. *Lectures in Mathematical Models of Turbulence*. Academic Press, London, England.
[17] Eiamsa-ard S, Ridlwan A, Somratyasin P, Promvonge P. 2008. Numerical investigation of turbulent flow through a circular orifice. *KMUTL Sci. J.* **8**(1), pp: 44-50.
[18] Çengel, Y. A., & Cimbala, J. M. 2006. *Fluid mechanics: fundamentals and applications*. 1980. On the calculation of turbulent heat transport downstream from an abrupt pipe expansion. *Numerical Heat Transfer*. **3**, pp: 189-207.
[19] White, F. M. 2011. *Fluid mechanics*. New York, N.Y., McGraw Hill.
[20] Versteeg, H. K., & Malalasekera, W. 2007. *An introduction to computational fluid dynamics: the finite volume method*. Harlow, England, Pearson Prentice Hall.
[21] Bruun HH, Khan MA, Al-Kayiem HH, Fardad AA. 1988. Velocity calibration relationships for hot-wire anemometry. *J Phys E: Sci Instrum*. **21**, pp: 225-232.