

Numerical Simulation of Swirl Decay in Turbulent Flow

Sukhada D. Kulkarni¹, P. N. Deshmukh², Ambrish D. Khot³Department of Mechanical Engineering, RMD Sinhgad School of Engineering, Warje, Pune, India^{1,2}Cummins India Ltd., Pune, India³

ABSTRACT: Swirling flows find their applications in combustion chambers of jet engines, turbo-machinery, mixing tanks, etc. Swirl is the vortices that can exist inside the cylinder. The single phase swirling flow in the straight pipe has been investigated since long. This investigation refers detailed experimental measurements using laser-Doppler anemometry (LDA) in a swirling flow through a straight pipe for wide range of Reynold No. This data is used for development of basic modelling approach using RNG K- ϵ model with standard and enhanced wall treatment available in commercial available tool like Fluent 16.2. Efforts are taken to improve the existing prediction by capturing swirl decay rate in better manner by modelling anisotropic turbulence. Another purpose of this study is to understand the swirl decay which is directly function of swirl number at inlet, the flow pattern induced by geometry, the distance from the pipe inlet, upstream turbulence data and nature of the inlet swirl profile. This helps to model swirl – wall interaction resulted velocity profile which leads to accurate predication of backpressure & turbulent mixing occurs in upstream of Diesel engine based after treatment devices.

KEYWORDS - Laser Doppler Anemometry, Reynold No., Mach No., swirl number, swirl decay, Numerical simulation, k- ϵ turbulence model, enhanced wall treatment

I. INTRODUCTION

Liquid or gas flow through pipes is commonly used in heating and cooling applications and fluid distribution networks. We pay attention towards the friction, which is directly related to the pressuredrop and head loss during flow through pipes. The pressure drop is then used to determine the pumping power requirement. Most fluids, especially liquids, are transported in circular pipe, because pipes with a circular cross section can withstand large pressure differences between the inside and outside without any significant distortion. Noncircular pipes are usually used in applications such as the heating and cooling systems. The fluid velocity in a pipe changes from zero to maximum from the surface because of the no-slip condition to the pipe center. In fluid flow, it is convenient to work with an average velocity V_{avg} . The velocity which remains constant in incompressible flow when the cross-sectional area of the pipe is remains constant. Practically we evaluate the fluid properties at some average temperature and treat them as constants. A careful inspection of flow in a pipe stats that the fluid flow is streamlined at low velocities but turns chaotic as the velocity is increased above a critical value. The flow region in the first case is said to be laminar, which characterized by the smoothstreamlines and highly ordered motion. Swirling flows used in a wide variety of engineering applications, such as in furnaces and gas turbine combustors. The usage of swirl in the power systems has several benefits. It is considered that a swirling flow produces an adverse pressure gradient that cause flow reversal or vortex breakdown. The central recirculation zone of swirling flow may results in decreasing pollutant emission by bringing hot species back to the combustion zone as well as lowering the possibility of flame blow-off. Moreover, swirling causes further mixing between the fuel and the oxidant. Much work has been done on swirling flow in straight pipe both experimentally and numerically. A common numerical finding of several researchers is that the Reynold's stress model is best suited for swirling flow than RANS eddy-viscosity models, such as the k- ϵ model. This is due to fact that the RSM handles anisotropy in the flow, whereas the k- ϵ model does not. This finding will important to present work, where a numerical simulation will be employed to better understand the physics of swirling pipe flow.

Prediction of turbulence and theoretical analysis has been made and to this date still is, the basic Fundamental problem of fluid dynamics, particularly in computational fluid dynamics (CFD). The major difficulty arises from the random

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

nature of turbulence phenomenon. Because of this unpredictability, it has been customary to work with the time averaged form of the governing equations, which results in terms involving higher order correlations of fluctuating quantities of the flow variables. The semi-empirical mathematical models used for calculating these unknown correlations form the basis for turbulence modelling. It is the focus of the present study to investigate the main principles of turbulence modelling and also includes examination of the physics of turbulence, closure model and application to specific flow conditions.

To replace the continuous problem domain with a discrete domain using a grid is the main strategy of the CFD. In the continuous domain, each flow variable is defined at every point. For instance, the pressure p in the continuous 1D domain shown in the below equation would be given as

$$p = p(x), \quad 0 < x < 1$$

Each flow variable is defined only at the grid point in the discrete domain. So, in this domain as shown below, the pressure defined only at the N grid points.

$$p_i = p(x_i), \quad i = 1, 2, \dots, N$$

In a CFD solution, one would directly solve for the relevant flow variable only at the grid points. The values at other locations are determined by interpolating the values at the grid point. The governing partial differential equations and boundary conditions are defined in terms of the continuous variables p , V etc., One can approximate these values in the discrete domain in terms of the discrete variables p_i , V_i etc. The discrete system is a large set of coupled, algebraic equations in the discrete variables. Setting up the discrete system and to solve it (which is a matrix inversion problem) involves a very large number of repetitive calculations, a task we humans palm over to the digital computer.

This idea can be extended to any general problem domain. The following figure shows the grid values used for solving the flow over an air foil. We'll take a closer look at this air foil grid soon while discussing the finite-volume method.

II. LITERATURE SURVEY

[1] 'Three-dimensional laser-Doppler velocimetry measurements in swirling pipe flow, Flow Turbulence Combust.' By Rocklage-Marliani et al, Flow Turbulence Combust. 70 (1-4) (2003) 43–67.

In this paper the evolution characteristics of swirl in a pipe for moderately high-Reynold numbers and over a wide range of swirl numbers have been measured which calculates the swirl strength. The non-intrusive method of three-dimensional laser-Doppler velocimetry used in obtaining measurements. Swirling flow introduced into the test section via a rotating tube bundle, which yield solid body rotation of the flow at the inlet. The evolution characteristics brought out dominant effect of swirling with a solid body rotation. The main aim of the authors to obtain "bench-mark" measurements from their experiment. Swirl was introduced by swirl generator into the test section with the constant angular velocity. The data used in the current research for comparison between the numerical and experimental results. A relation between experimental and theoretical is desirable, so the experimental design measures the flow measurements by considering the errors and considering relevant initial and boundary conditions. The density ρ and viscosity μ of this working fluid are 835 kg/m^3 and $1.07 \times 10^{-3} \text{ Pas}$ respectively at a temperature of 20°C (293 K). Since these fluid properties are so close to that of water, Reynolds numbers of the fluid flow is 2.8×10^5 based on the pipe diameter. These input parameters are considered for the fluent simulation which combines the theory and experiment results. By considering these inputs, simulation results of axial velocity and azimuthal velocity results were plotted. In experimental analysis, angular velocity equivalent to lower the errors in the evaluation of the axial vorticity. The experiment considered the angular velocity distribution corresponding to the solid body rotation. By the observation conclusion made that the helical streamline flow either axisymmetric or cellular possess the angular velocity distribution at the core region. By which the swirl at the fluctuation motion can be defined which reduces the Reynold shear stress effects. Swirl regarded as the flow 'laminarise' by considering the tangential motion.

[2]'Comparison of turbulence models in simulating swirling pipe flows' By Andrew Escue, Jie Cui, Applied Mathematical Modelling 34 (2010) 2840–2849

Swirling in the flow is a common feature in the many engineering applications. A numerical study carried out on the swirling flow in a straight pipe by using the commercial CFD Fluent. For this study two turbulence models were used

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

as Reynold stress and the RNG $k-\epsilon$ with the performance of the two dimensional simulations. By considering various swirl numbers results were obtained and compared the results with the experimental data to validate the numerical method.

The objective of this paper is to use the CFD Fluent model to develop the swirling flow in a straight pipe and to obtain the turbulence model which is involved in the simulation. The results are obtained by solving 'Navier- Stokes equations' and compares with the experimental results present in the literature.

The experimental test section made from boro-silicate glass of a 1500 mm long pipe with inner diameter 100 mm. The data was for a nominal Reynolds number of 2.8×10^5 on the pipe diameter D and the mean axial velocity U_{ave} .

[3] 'LDA measurement in a strongly swirling flowing a pipe' By B. Z. Genç, J. Jovanović, and F. Durst 6. – 8. September 2005, BTU Cottbus

The main objective of this paper is to measure turbulence statistics. The experimental data obtained for the mean of the objective of this paper is to present the experimental facility produced for the investigation of swirling pipe flows in LSTM-Erlangen and the implementation of the LDA measuring was discussed. Technique velocity and second-order turbulence statistics in the axial flow direction will be presented.

Measurements were carried in detail using laser-Doppler anemometry (LDA) in a swirling flow through a pipe for the flow rate which corresponds to the Reynold number. The profiles for velocity and statistical moments yield measured insight into the behaviour of turbulence subjected to swirl. The swirl decay and the corresponding change in the turbulence structures was investigated in the entrance region, up to approximately 20 diameters at the downstream of the pipe.

This study resulted in the swirling effect on the mean velocity and RMS value. Study concluded that, the net effect of a swirling nothing but the transfer of the momentum from the axial direction to the radial direction at any of the swirl rate due to the centrifugal effects. The reduced vortex core energy transmitted to the small scale turbulent fluctuation due to the dissipation of the mechanism.

[4] 'Analytical velocity profile in tube for laminar and turbulent flow' By Jaroslav Štigler, Engineering MECHANICS, Vol. 21, 2014, No. 6, p. 371–379

In this article for both laminar and turbulent velocity profiles formula with circular cross section is to be developed. This formula is rather simple and easy to use. The main advantage of this velocity profile is that same formula can be used for laminar and turbulent flow. The new formula compared with power law velocity profile and with the law of the wall called as the log-law. The boundary layer nothing but the near wall region where the frictional forces and dynamic forces are to be compared

In power law velocity profile two fundamental discrepancies are observed are as follow-

1. One appear near the wall of tube, it contains infinite shear stress on wall which represents infinite friction losses in tube.
2. Another one occurred at the velocity profile smoothness at the centre of the tube.
3. By using empirical, formulas problems are to be solved the near wall problems by different ways. Generally the boundary layer is divided into the three parts are as follows-
4. Viscous sub-layer
5. Transition area
6. Turbulent boundary layer

Velocity profile based on the vorticity density γ distribution on the tube cross section, which is related to the vorticity vector Ω . the velocity profiles obtained from the both methods are compared, at the infinite derivative power law velocity profiles suffered some problems near the wall.

New velocity profiles were developed which resolves all the problems of the power law velocity profile, which describes the velocity profile for both laminar flow and the infinite Reynold number.

The new profile introduces the interesting features near the wall velocity profile. Again one problem is arrived that infinite curvature radius. This problem can be overcome in the future studies.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

[5] 'Assessment of the Performances of RANS Models for Simulating Swirling Flows in a Can-Combustor' By K. KhademiShamami and M. Birouk, *The Open Aerospace Engineering Journal*, 2008, 1, 8-27

The paper presents the performances of RANS turbulence model for simulating turbulent swirling can-combustor flows with different inlet swirl intensities assessment. The predictions were compared against published experimental data reveal that the eddy-viscosity models cannot show the central recirculating zone in the case of a weakly swirling flow. By using the realizable $k-\epsilon$ and the SST $k-\epsilon$, the predictions of turbulence intensities comparable to those of the Reynolds stress closures. The shear stresses were not predicted well by all the tested models. Both the eddy-viscosity and the Reynolds stress closures shows the less approximation errors in the weakly swirling flow. Different Reynolds stress model versions are tested by considering different swirling flow.

By experimental results it was concluded that the swirl intensity can change the swirl flow field characteristics, so that for the analysis RANS turbulence model is used. In the study, mean and fluctuating velocity components and Reynolds shear stresses are compared with the literature result.

By using combination of RSM and SSG models gives the good predictions related to the axial velocity profile and the tangential velocity profile at the near flow field region in a combustor.

Study concluded some result which are as follows-

1. The RSM and SSG models gives the accurate mean velocity profiles at the weak swirl numbers whereas these models are unable to give the accurate results of tangential velocity profiles at the high swirl number.
2. All the turbulence models employed gives the accurate predictions to the effect of the swirl intensity at the axial direction of flow reattachment point at the corner.
3. CTRZ can be predicted by all models in the strong swirling flow condition but in weak swirling flow condition only RSM and SSG models are capable of predicating CRTZ.
4. At the anisotropic flow numerically calculated stress for the turbulence quantity is more accurate than the eddy viscosity model result.

[6] 'Evaluation of numerical schemes using different simulation method for the continuous phase modelling of cyclone separators' By Sujeet Kumar Shukla, PrashantShuklaPradyumnaGhosh, *Advanced Powder Technology* 22 (2011) 209-219

The RSM turbulence model revisited to explore its simulation capability of fluctuating velocity profiles and PVC phenomenon of cyclone separator. The predicted mean, pressure drop and fluctuating velocity profiles inside the cyclone separator with steady and unsteady simulations were compared to experimental results available in literature. In the results the low swirl number flow configuration is carried out. Normalised axial velocity fluent results are compared with the presented paper result. In this study three regions are to be studied are as-

- (i) A core region near the centreline of the combustor,
- (ii) A near wall region,
- (iii) A mixing layer between these two regions.

Similarly Normalised tangential velocity fluent results are also compared with the paper data. For several stations which are in axial direction represents the near and mid as well as far-fields of the flow development in the combustor. By studying cyclone separators simulation results of the performance of different numerical schemes under the steady and unsteady conditions. The pressure drop, static pressure drop distribution profile and mean velocity profile predictions by unsteady simulation are closer to the experimental values compare to the predictions by the steady simulation. In core and off both regions the experimental data of fluctuating velocity profiles was simulate successfully by high order upwind scheme.

It is concluded by the simulation result is that large eddies in core region can be resolved by using RSM turbulence model with PVC phenomenon.

[7] 'Study of barrel swirl in a four-valve optical IC engine using Particle Image velocimetry' By M. Reeves, C.P. Garner, J. C. Dent and N.A. Halliwell, *International Synopsium COMODIA 94* (1994)

In a production geometry swirl barrel characterised by using (PIV) Particle Image Velocimetry through the compression stroke. The images have been recorded by the PIV at the crank angle in parallel plane to the cylinder axis through compression. This can be happen by developing of a special corrective optical system which gives diffraction limited imaging of particles in the thick glass cylinder.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

The LDA measurements were carried out on a coarse grid which reveals the single large scale tumble vortex direction into the anticlockwise.

As per the paper study, major feature noticed that the compression velocity field is similar to the parallel plane through the spark with velocity 11m/s at a speed of 1000 RPM. Paper study suggest that by modifying high speed the flame kernel growth can be achieved. The scattering efficiency such as the illumination energy can be improved by using slightly larger seeding particles. Which reduces the drop out data obtained by flare and stray reflection. The experiment shows that the use of adaptive thresholding techniques which results in the rejection of film noise produced by interrogation regions. Which improves the yield data and allow the velocity measurements in the one millimetre from the combustion chamber wall.

PIV technique is used to study the evolution of barrel swirl in the realistic geometry pent roof of four valve motored engine. The study carried out from BDC to ignition point. Same study is carried out by using Hotwire and Particle Tracking Measurements on the same geometry. By plotting Cross correlation following results are obtained as-

1. PIV performance improve
2. Directional ambiguity resolve
3. Dynamic range expand
4. Signal to noise ratio improve
5. Spatial resolution improve

III. EXPERIMENTAL SETUP AND ANALYSIS

By taking the reference for the experimental data from the ROCKLAGE-MARLIANI paper. The same model we used in this paper for the validation and the calculation of the swirl decay.

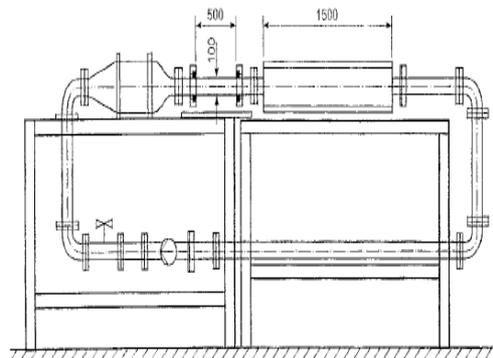


Fig No. 1 LDA Experimental setup

The facility is basically a closed-loop pipe flow installation driven by a screw conveyer pump operating in suction mode. The diameter D and the length L of the pipe were 100 mm and about 1500 mm respectively. A swirl generator, which is basically a piece of was installed just before the inlet of the pipe. So this device having a honeycomb structure (see Fig. 1) and mounted on two bearings, was used to create swirling flows of desired strengths when rotated at different angular speeds. The swirl generator was rotated by using a three-phase AC motor controlled by a frequency converter. To provide optical access into the pipe for LDA measurements and to enable refractive index matching, the walls of the rectangular test section were made of the same glass material (Duran-50 glass) as the pipe wall. Therefore, when the test section is filled with the working fluid (cumen-paraffin) with density 835 kg/m^3 , viscosity $1.07\text{e-}03 \text{ kg/m.s}$, Reynold No. 2.8 E5, it provides an optically uniform medium which enables the laser light to penetrate through the pipe without any deflection. In addition to this, a fluid temperature control system was employed to eliminate the thermal effects on the refractive index of the fluid and keep the index constant within a certain degree of accuracy.[3]

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

Pressure Drop (Laminar) -

$$\Delta P = \frac{8\mu L Q}{\pi r^4}$$

Here –

ΔP = the pressure loss in Pa

μ = the dynamic viscosity in kg/m.s

Q = the volumetric flow rate in m/s

r = the radius in m

π = Mathematical constant Pi

L = the length of pipe in m

v = The velocity in m/s

By this calculation pressure drop is obtained as-

Pressure Drop (Turbulent)-

$$\Delta p = f_D * \frac{L}{D} * \frac{\rho v^2}{2}$$

Where-

ΔP = The pressure loss in Pa

f = The Darcy friction factor = 0.03 (from moody chart)

L = Length of pipe or pipe part in m

D = Diameter of the pipe in m

ρ = Density of fluid in kg/m³

V = Flow velocity m/s

Mass Flow Rate

The calculation of the Reynolds number, and the values where laminar flow occurs, will depend on the geometry of the flow system and flow pattern. Air and cummen-paraffin are used as a working fluid in this project so dynamic viscosity and density of the air and cummen-paraffin for room temp can be obtained from standard tables. According to the Reynolds no. mass flow rate can be calculated as-

$$Re = \frac{4m}{\pi D \mu}$$

Here,

Re = Reynolds No.

m = Mass flow rate in kg/s

D = Diameter of geometry in m

π = Mathematical constant Pi

μ = Dynamic Viscosity of working fluid in kg/m.s

- Analysis

Laminar and Turbulent flow calculation (CFD Simulation) –

- Geometry-

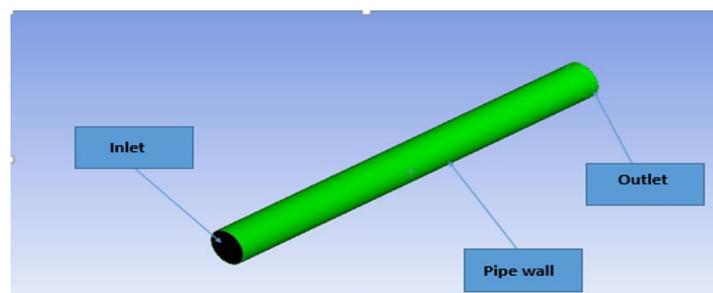


Fig. 2 Actual Geometry with named parts (Inlet, outlet, pipe wall)

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

A simple straight pipe with following Specification

Length = 1.5 m

Diameter = 0.1 m

- **Meshing –**

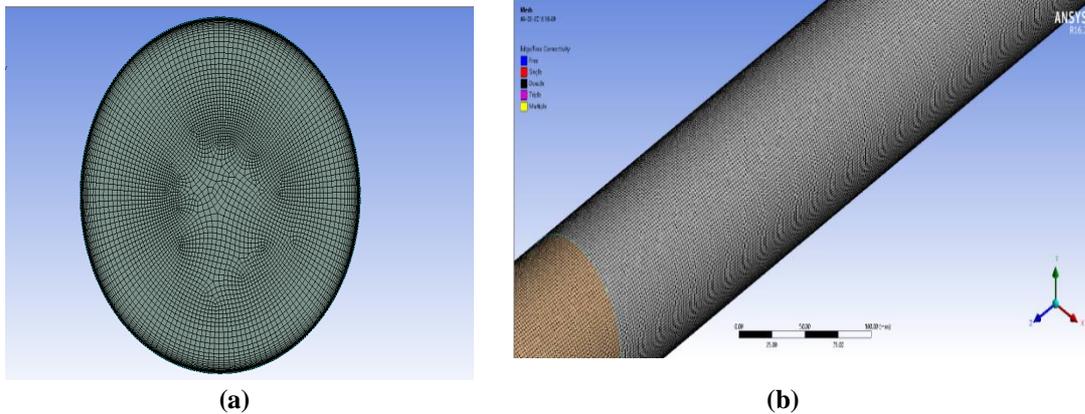


Fig : (3a) Meshing across the diameter of the pipe
(3b) Meshing along the length of the pipe

- **FLUENT settings (Laminar Flow)**

- **Boundary conditions:**

Inlet mass flow rate (\dot{m}) = 0.00219 kg/s

Temperature = 300 K

- **Modelling assumptions**

Flow is Laminar flow so using laminar model In ANSYS fluent 16.2

VI. SIMULATION RESULT

- **Results Obtained –**

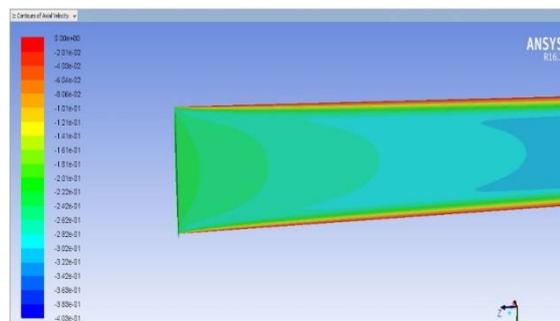


Fig.(4a) The fully developed profile region for laminar flow

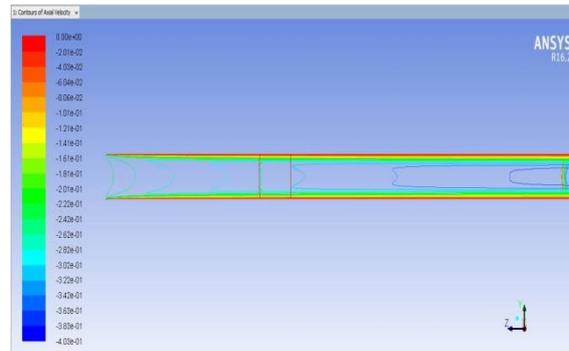


Fig. (4b) The fully developed profile region for laminar flow

To obtain the results in a pipe it is necessary to calculate the hydrodynamic length which shows the fully developed profile. For Laminar flow Hydrodynamic length can be calculated as –

$$0.05 \cdot Re \cdot D$$

Here

Re = Reynold no.

D = Diameter in m

Hydrodynamic length obtained from this calculation is **0.75m**.

According to the previous calculation length for one plane is created along the z axis as shown in fig.

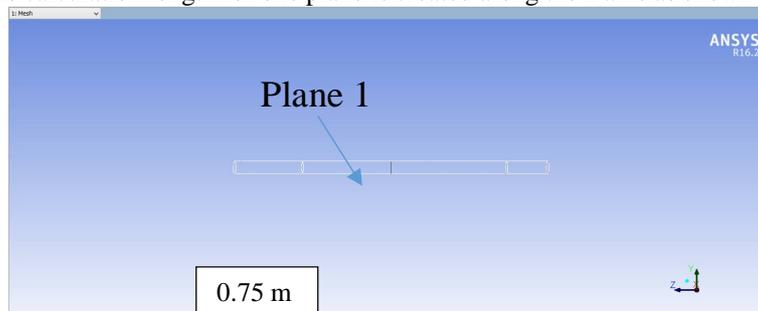


Fig. 5 Plane creation

- According to plane 1 Velocity profile obtained -

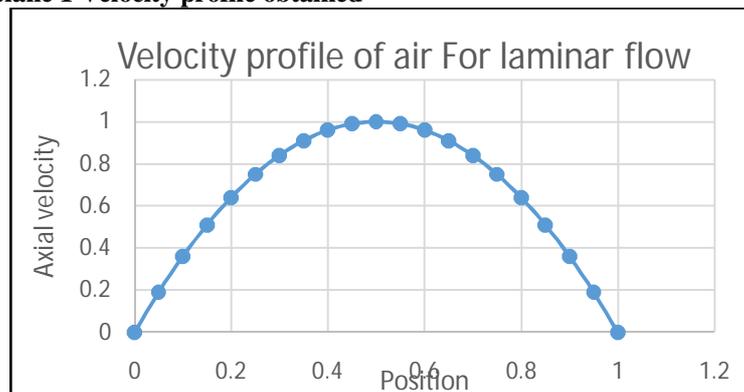


Fig. 6 fully developed profile along the plane

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

- **FLUENT** settings (Turbulent Flow)
- **Boundary conditions:**
 - Inlet mass flow rate (\dot{m}) = 0.0117 kg/s
 - Temperature = 300 K
 - Pipe Diameter = 0.1 m
 - Intensity = 5 %
- **Modelling assumptions**
 - Flow is Steady-state turbulent flow
 - Turbulent model are
 1. Realizable k-epsilon with Standard wall treatment ($Y^+ < 10$ is maintained throughout the domain)
 - Ideal Gas for density.
- **Results obtained-**

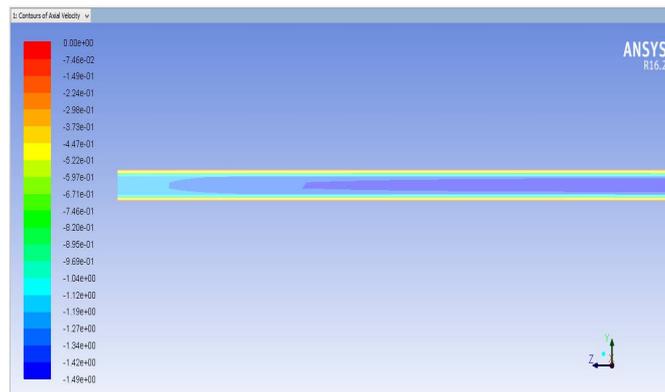


Fig.7 (a) The fully developed profile region for turbulent flow

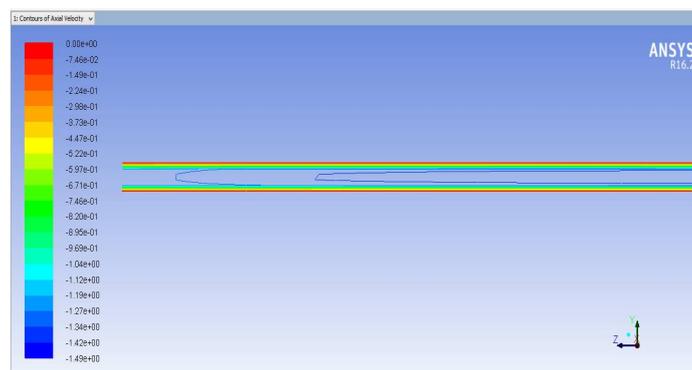


Fig.7 (b) The fully developed profile region for turbulent flow

To obtain the results in a pipe it is necessary to calculate the hydrodynamic length which shows the fully developed profile. For Turbulent flow Hydrodynamic length can be calculated as –

$$1.359 * D * Re^{(1/4)}$$

Here-

Re = Reynold no.

D = Diameter in m

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

Hydrodynamic length obtained from this calculation is 0.75m.
According to this length one plane is created as shown in fig. 8

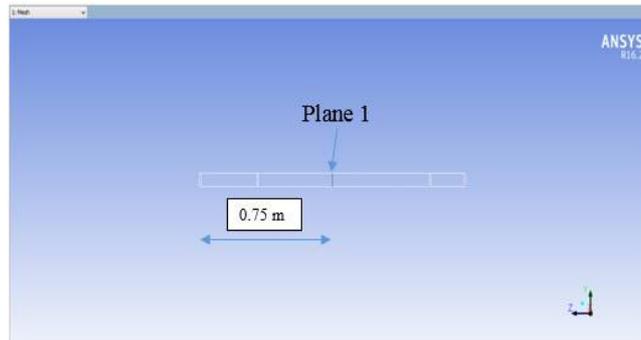


Fig. 8 Plane creation

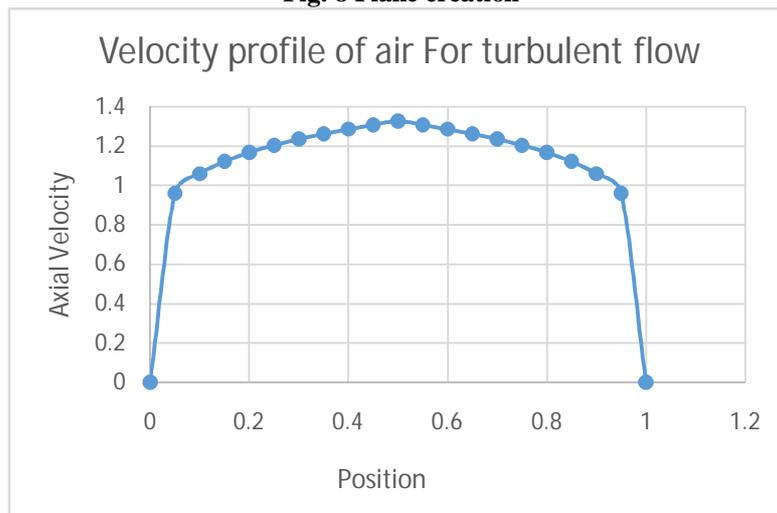


Fig. 9 fully developed profile along the plan

- Laminar Flow

By the simulation calculation pressure drop is obtained as-

For Re (1500) = $\Delta P = 0.025$ Pa

- Turbulent flow

By the simulation calculation pressure drop is obtained as-

For Re (8000) = $\Delta P = 0.516$ Pa

- **Error result**
- **Laminar flow**

In the laminar flow the pressure drop is calculated by using Hagen-poiseuille equation, as we see earlier in equation no. from the equation we get pressure drop as **0.021 Pa**, and from the simulation we get pressure drop as **0.025 Pa**

By considering Calculation result and simulation result there is some error up to 0.4 %. Which is occur due to the meshing method. But in industry up to 5 % error is acceptable so I consider these results for the further study

- **Turbulent flow**

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

In the laminar flow the pressure drop is calculated by using Darcy's equation, as we see earlier in equation no. from the equation we get pressure drop as **0.514 Pa**, and from the simulation we get pressure drop as **0.516 Pa**. By considering Calculation result and simulation result there is some error up to 0.4 %. Which is occur due to the meshing method. But in industry up to 5 % error is acceptable so I consider these results for the further study

Velocity Profile calculation for Laminar flow-

The laminar velocity profile of the fluid flow governed by the pressure gradient is parabolic. This expression is more suitable for the practical use because the average velocity can be easily expressed from the flow rate and the radius of tube.

$$u(r) = 2 V_{avg} \left(1 - \frac{r^2}{R^2}\right)$$

Here-

V_{avg} = Average velocity in m/s

R = Radius of the pipe in m

r = Distance from the wall in m

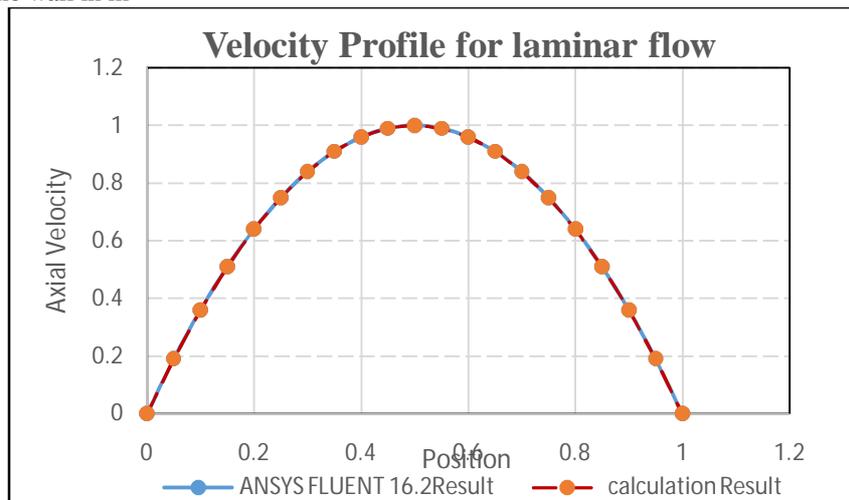


Fig.10 Velocity profile obtained by calculation

Velocity Profile calculation for Turbulent flow-

In many engineering calculations it is usual to employ a power law velocity profile given as for the turbulent flow given as-

$$\frac{u}{U_{max}} = \left(1 - \frac{r}{R}\right)^{1/n}$$

Here –

U_{max} = Max. Velocity in m/s

R = Radius of the pipe in m

r = Distance from the wall in m

The exponent "n"=A function of Reynolds Number. However, (n=7 seems to be applicable to a wide range of pipe flows and is the one commonly used)

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

- **Results**

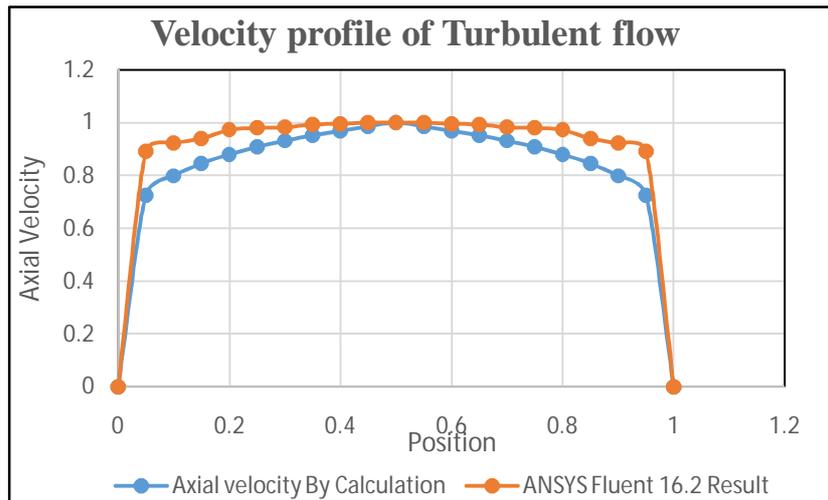


Fig. 11 Velocity profile obtained by calculation

After plotting the graphs 0.4% error is observed between the velocity profiles obtained by fluent 16.2 result and calculation result for the turbulent profile

Velocity Profile Angle for Laminar flow

Considering first point on the both velocity profile/curve the velocity angle can be calculated. Velocity angles for laminar flow obtained after calculation are-

- Angle obtained on calculation & Power law result profile = 14.7°
- Angle obtained on simulation result profile = 14.7°

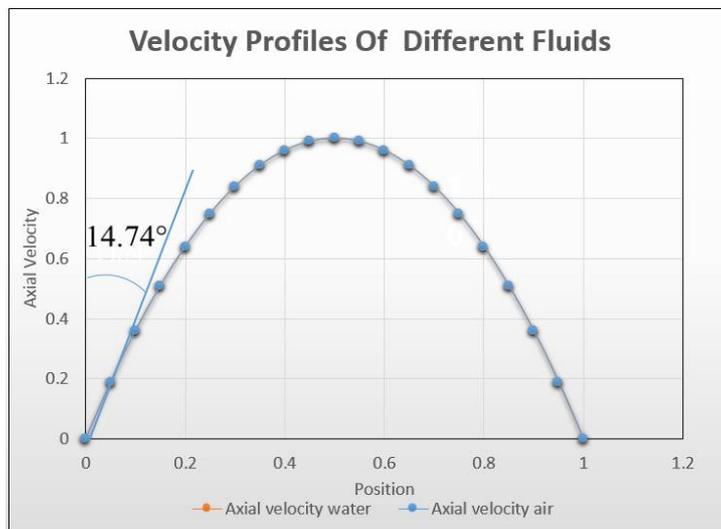


Fig. 12 Velocity profile Angle For Laminar Flow

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

Velocity profile for laminar flow using different Reynold No. –

By considering different Reynolds number to study the change in the velocity profiles of laminar flow and also study the change in the angle created by the respective velocity profile with the pipe wall. For the study we consider the three Reynolds Number as 500 , 1000, 1500. With the earlier same boundary conditions setup , just to change the mass flow rate of the respective Reynolds number and run the simulation.

The Simulation result obtain are shown in fig no. 13

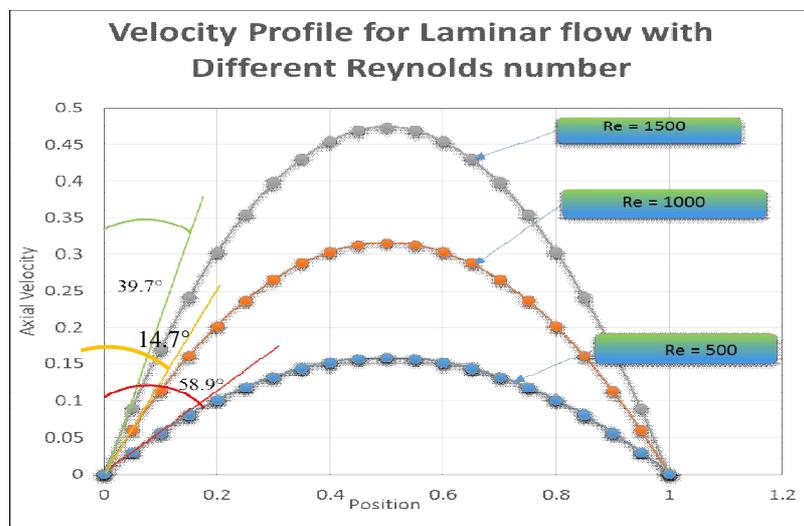


Fig. No. 13 Velocity profile with Different Reynold No. for laminar flow

Maximum Axial Velocities obtain at respective profiles are as follows.-

- Maximum Axial velocity at Re. 500 = 0.165 m/s
- Maximum Axial velocity at Re. 1000 = 0.315 m/s
- Maximum Axial velocity at Re. 1500 = 0.475 m/s

Results shows that , as the Reynold no. increases the frictional velocity also increases with decrease in angle.

Velocity Profile Angle for Turbulent flow

Considering first point on the both velocity profile/curve the velocity angle can be calculated. Velocity angles for laminar flow obtained after calculation are-

- Angle obtained on calculation result profile = 3.9°
- Angle obtained on simulation result profile = 3.2°

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

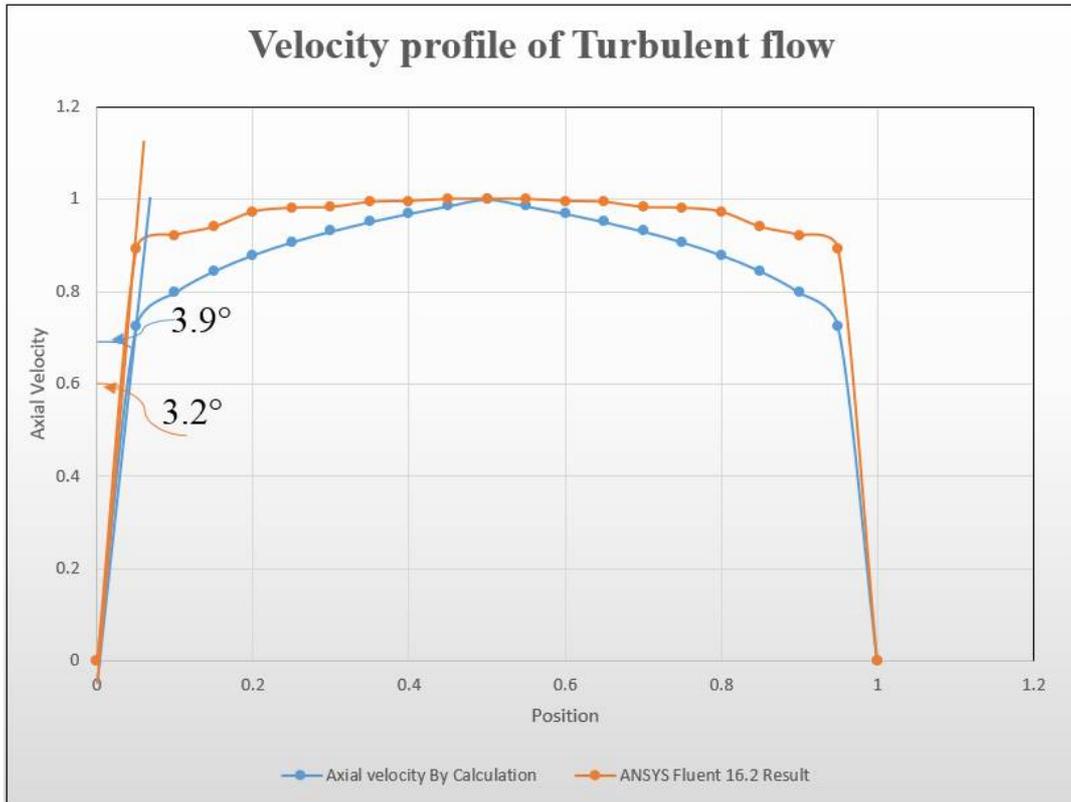


Fig. 14 Velocity profile Angle for Turbulent Flow

Velocity profile for turbulent flow using different Reynold No.

By considering different Reynolds number to study the change in the velocity profiles of turbulent flow and also study the change in the angle created by the respective velocity profile with the pipe wall.

For the study we consider the four Reynolds Number as 8000, 10000, 75000, 100000. Apply the k-ε turbulence model for simulation, change the mass flow rate of the respective Reynolds number and run the simulation.

The Simulation result obtain are shown in fig no.14

U⁺ vs Y⁺ Graph-

U⁺ vs Y⁺ graph plotting is important to calculate the average velocity of the turbulent flow at the certain location within the fluid region. For plotting this graph 'law of the wall' is used. Law of the wall gives the proportionality between turbulent velocity and the logarithmic distance from the point within the boundary region. U⁺ and Y⁺ values calculated numerically with some constant values as-

$$u^+ = \frac{1}{K} \ln y^+ + C^+$$

$$y^+ = \frac{y u_t}{\nu}$$

$$u_t = \sqrt{\frac{\tau}{\rho}}$$

$$u^+ = \frac{u}{u_t}$$

$$u = \frac{u_t}{K} \ln \frac{y}{y_0}$$

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

Here,

y^+ = Wall coordinate

u^+ = Dimensionless velocity

τ = Wall shear stress

K = Von Karman constant

ln = Natural logarithm

y = Distance from the boundary wall

y_0 = Distance from boundary at which idealized velocity is zero

u = Velocity

u_t = Friction velocity

ρ = Density

C^+ = Constant

For the smooth wall the constant values are –

K = 0.41

$C^+ = 5$

By keeping the simulation setup for two turbulence models as k- ϵ model and k - ω model, to identified that which gives the better result. From the simulation u_t and τ values were obtained. By using these values in above formulas the values of U^+ and Y^+ calculated and graph is plotted by using these results. In the graph the U^+ vs Y^+ values plotted with respect to the two graphs $U^+ = Y^+$ and $u^+ = 1/K \ln y^+ + C^+$ Which shows in fig. 7.15 and 7.16 respectively for k- ϵ model and k- ω model.

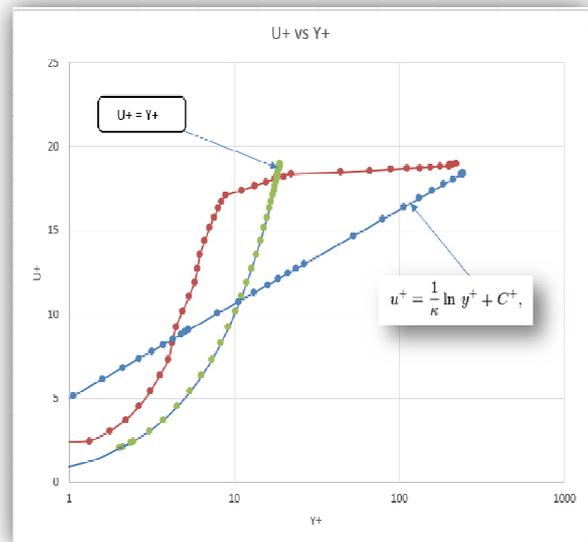
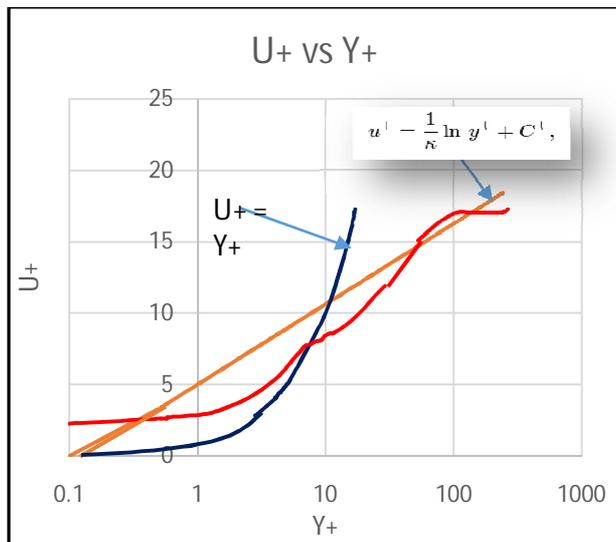


Fig. No 15 U^+ vs Y^+ graph (k- ϵ turbulence model) Fig. 16 No U^+ vs Y^+ graph (k- ω turbulence model)

From the above graphs we can see that, by using k- ϵ turbulence model the U^+ vs Y^+ curve is within the range of $U^+ = Y^+$ and law of wall (u^+) compare to the k- ω model, so we can say that the k- ϵ model is best suited for simulation of the turbulence flow which gives the results within the acceptable range.

VIII. CONCLUSION

- Good agreement between predicted and experimental results was obtained for u, v, w velocity values at different locations post the swirl and control pipe. Despite the CFD model respectively under- and over-predicting tangential velocities.
- CFD predictions still followed the same trend as the experimental results and the predicted swirl decay were in accordance with those resulting from the experiments.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 6, June 2016

- The models accurately predicted the trend of the gauge static pressure results when compared to experimental results, but slightly over predicted the values. Visualization experiments with the high speed camera, PIV and LDA measurements and numerical predictions all proved that the swirl pipe was effectively inducing swirl to the lean phase pneumatic flow.
- A decaying swirl pattern was formed in the same direction as the twists in the swirl pipe geometry, which is non-existent when the control pipe was used. Axial velocities were minimum at the pipe wall and maximum at the centre of the pipe. In contrast, tangential velocity is minimum at the centre of the pipe with a sudden increase

REFERENCES

- [1] Bianco, F., Chibbaro, S., Marchioli, C., Salvetti, M.V., Soldati, A., 2012. Intrinsic filtering errors of Lagrangian particle tracking in LES flow fields. *Phys. Fluids* 24,045103-1
- [2] Chibbaro, S., Minier, J.-P., 2008. Langevin PDF simulation of particle deposition in a turbulent pipe flow. *J. Aerosol Sci.* 39, 555–571
- [3] Chung, J.N., Troutt, T.R., 1988. Simulation of particle dispersion in an axisymmetric jet. *J. Fluid Mech.* 186, 199–222
- [4] Cousins, L.B., Hewitt, G.F., 1968. Liquid Phase Mass Transfer in Annular Two-Phase Flow. UKAEA Report. AERE-R 5657.
- [5] Crowe, C., Sommerfeld, M., Tsuji, Y., 1998. *Multiphase Flows with Droplets and Particles*. CRC Press, New York
- [6] Delfos, R., Murphy, S., Stanbridge, D., Olujic, Z., Jansens, P.J., 2004. A design tool for optimising axial liquid–liquid hydrocyclones. *Min. Eng.* 17, 721–731
- [7] Eggels, J.G.M., 1994. Ph.D. Thesis. Delft University of Technology, The Netherlands. Eggels, J.G.M., Unger, F., Weiss, M.H., Westerweel, J., Adrian, R.J., Friedrich, R., Nieuwstadt, F.T.M., 1994. Fully developed turbulent pipe flow: a comparison between direct numerical simulation and experiment. *J. Fluid Mech.* 286, 175–209
- [8] Fukagata, K., Kasagi, N., 2002. Highly energy-conservative finite-difference method for the cylindrical coordinate system. *J. Comput. Phys.* 181, 478–498
- [9] Gomez, L., Mohan, R., Shoham, O., 2004. Swirling gas–liquid two-phase flow – experiment and modeling – Part I: swirling flow field. *J. Fluid Eng. – T ASME* 126, 935–942
- [10] Kitoh, O., 1991. Experimental study of turbulent swirling flow in a straight pipe. *J. Fluid Mech.* 225, 445–479 Klujso, L.A., Rafaelof, M., Rajamani, R.K., 1999. Dust collection performance of a swirl cleaner. *Powd. Technol.* 103, 130–138
- [11] Kuerten, J.G.M., van Kemenade, H.P., Brouwers, J.J.H., 2005. Numerical study of the rotational phase separator sealing impeller. *Powd. Technol.* 154, 73–82
- [12] Lund, T.S., Wu, X., Squires, K.D., 1998. Generation of turbulent inflow data for spatially-developing boundary layer simulations. *J. Comput. Phys.* 140, 233–258
- [13] Marchioli, C., Soldati, A., 2002. Mechanisms for particle transfer and segregation in turbulent boundary layer. *J. Fluid Mech.* 468, 283–315
- [14] Marchioli, C., Giusti, A., Salvetti, M.V., Soldati, A., 2003. Direct numerical simulation of particle wall transfer and deposition in upward turbulent pipe flow. *Int. J. Multiphase Flow* 29, 1017–1038
- [15] Marchioli, C., Salvetti, M.V., Soldati, A., 2008a. Some issues concerning large-eddy simulation of inertial particle dispersion in turbulent bounded flows. *Phys. Fluids* 20, 040603-1
- [16] Marchioli, C., Soldati, A., Kuerten, J.G.M., Arcen, B., Taniere, A., Goldensohn, G., Squires, K.D., Cargnelutti, M.F., Portela, L.M., 2008b. Statistics of particle dispersion in direct numerical simulations of wall-bounded turbulence: results of an international collaborative benchmark test. *Int. J. Multiphase Flow* 34, 879–893
- [17] Muntean, S., Ruprecht, A., Susan-Resiga, R., 2005. A Numerical Investigation of the 3D Swirling Flow in a Pipe with Constant Diameter. Part 2: Turbulent Computation. *Scientific Bulletin of the Politehnica University of Timisoara, Transactions on Mech.* Nieuwstadt, F.T.M., Dirkszager, M., 1995. A fluid mechanics model for an axial cyclone separator. *Ind. Eng. Chem. Res.* 34, 3399–3404
- [18] Nygard, F., Andersson, H.I., 2010. DNS of swirling turbulent pipe flow. *Int. J. Numer. Meth. Fluids* 64, 945–972