SIMULATION OF FLOW THROUGH MICROTUBE

[redacted]
jouhararahman6@gmail.com
[redacted]

[redacted]
saji@gectcr.ac.in
[redacted]

ABSTRACT

The development of flow through microtube and the evaporation from that would be of interest in many fields. The current study aims at developing a CFD model for the simulation of flow through micro tubes and the evaporation from its end. The process in the micro tube is manifested by the action of surface tension.

Capillary rise dynamics in a 0.2 mm diameter tube is first studied by simulating the flow using CFD (ANSYS FLUENT software). The model is validated with published experimental data. Further this paper report the CFD simulations performed to simulate the combined effect of capillary rise and evaporation. Two dimensional model using volume of fluid method is adopted for the simulation of capillary rise along with evaporation. It is studied whether such a modeling exercise using CFD model representing combined effect of capillary rise and evaporation can yield reasonable practical results. Good agreement between the CFD results and experimental data has been observed.

KEYWORDS

Capillary rise, evaporation, surface tension, microtube.

1.0 INTRODUCTION

Microtubes are the tubes that have inner diameters as a few micrometers. The flow through microtubes and the evaporation from that would be of interest in many fields. The channels in soil through which water moves is of microdimension. The most distinctive xylem cells are the long tracheary elements that transport water is of microdimension. The analysis of flow through such tubes is difficult from the point of view of accuracy. Analysis of the process of evaporation from such microdimensioned tubes also a complex process owing to involvement of conservation of energy. This paper describes the numerical simulation of flow through microtubes followed by evaporation using ANSYS FLUENT software.

2.0 LITERATURE REVIEW

An experimental study conducted by Zhmud (2000)[1] gives an overview and detailed analysis of the classical theory of capillarity. A number of known equations of capillary rise dynamics are available which are limited by many assumptions. The role of nonlinear dissipation and flow pattern effects near the capillary entrance are discussed by them. Surfactant solutions got special attention in the study. A simplified relation for the capillary rise dynamics in the case of strong depletion of the interfacial region is also proposed by them.

A study conducted by Richards (1931)[2] gives a brief overview of the capillary conduction of liquids through porous mediums. The flow of liquids in unsaturated porous mediums follows the ordinary laws of hydrodynamics as the motion is produced by gravity and the pressure gradient force acting in the liquid. By making use of Darcy's law an equation is derived for the capillary conduction of liquids in porous mediums by him. The possible existence of a hysteresis effect between the capillary potential and moisture content of a porous medium is also taken into consideration in the study.

Lu and Likos (2004)[3] studied the rate of capillary rise in soil by the closed-form analytical solution developed. The new developed solution makes significant improvement over the Terzaghi's classical solution by ignoring the nonlinearity of hydraulic conductivity with respect to changing soil suction. It is reported that the new equation developed in the study is more realistic in practical predictions for the rate of capillary rise in unsaturated soils.

Cazadero (2004)[4] describes well about the capillary action in trees. It is well explained how trees can get water all the time from its bottom roots to the top leaves. It is due to the action of capillary effect and osmosis.

Vasu and Wahid (1990)[5] give the transpiration rate and biomass of coconut palm which have been found out using tritiated water as a tracer. The method of tracer injection into the coconut trunk and the extraction of tritiated water from coconut leaves are outlined. The transpiration rate of the tree selected for the study was found to be 2.2 litres/hour with a total biomass of 172 kg. Their result gives an idea about the rate of transpiration in coconut trees.

Zhang et al. (2009)[6] monitored the liquid flow through microtubes using a micropressure sensor. The pressure driven liquid flow through microtubes was studied by them in a range of very low Reynold's numbers (<0.15) by monitoring the pressure change in situ. A good linear relation for the pressure drop versus flow rate was observed in the study.

A study conducted on Numerical simulation of flow through microchannels with designed roughness by Rawool et al. (2006)[7] came up with the results that demonstrate the effect of roughness height (surface roughness), geometry, Reynolds number on the friction factor. The study concludes that the friction factor is more for rectangular and triangular obstructions and it decreases as the obstruction geometry is changed to trapezoidal shaped.

Though there are many studies on flow through microtubes the studies on the numerical simulation of dynamics of flow in the microtubes are limited. Also the studies on the evaporation from microtubes are limited, though it has application as in the case of water transport in trees. Hence an attempt is made to numerically simulate the capillary rise dynamics and evaporation from the microtubes.

3.0 CONVENTIONAL EQUATIONS FOR CAPILLARY RISE

Capillarity is the ability of liquids to penetrate into fine pores and cracks with wettable walls and be displaced from those with non-wettable walls[1]. Several equations have been being developed to express the relationship between capillary rise and time. Lucas-Washburn equation and Bosanquet equation are two conventional equations among them.

The Lucas-Washburn equation describes capillary flow in a bundle of parallel cylindrical tubes. The simplified form of Lucas-Washburn equation is given by the equation (1),

$$L^2 = \frac{\Upsilon D t \cos\Phi}{4\eta} \qquad (1)$$

Where, L= Liquid penetration distance

ϒ= Liquid-air surface tension.

η= Dynamic viscosity of liquid.

t= Time of penetration.

D= Pore diameter.

Φ= Contact angle between liquid and solid

In the Lucas-Washburn equation the inertia of the fluid is ignored leading to an unphysical infinite initial velocity of fluid motion. An improved modification of the Lucas-Washburn theory is Bosanquet equation, which is a differential equation that is second order in the time derivative and therefore fluid inertia is taken into account. It is assumed that the motion is completely driven by surface tension, with no applied pressure at either end of the tube.The Bosanquet equation of motion is expressed by equation (2),

$$\frac{d}{dt}\left(\pi r^2 \rho x \frac{dx}{dt}\right) + 8\pi\eta x \frac{dx}{dt} = 2\pi r \Upsilon \cos\theta \quad (2)$$

Where, r= Radius of capillary.

η= Dynamic viscosity of liquid.

ρ= Fluid density.

x= Distance the fluid advanced.

ϒ= Liquid-air surface tension.

θ= Contact angle between liquid and solid.

The solution of Bosanquet equation is expressed by the equation (3),

$$x(t)^2 - x(0)^2 = \frac{2b}{a}[t - \frac{1}{a}(1 - e^{-at}) \quad (3)$$

Where , $a = \frac{8\eta}{\rho\, r^2}$

and $b = \frac{2\Upsilon\cos\theta}{\rho r}$

Unlike the solution of Washburn's equation the function x(t) does not have an unphysical infinite time derivative at t=0. The analysis of capillary rise with respect to time can be done either by experimental method or by numerical techniques. Experimental method needs assistance of high precision cameras for observation.

4.0 NUMERICAL SIMULATION

The technical advancement in the field of Computational Fluid Dynamics (CFD) can be used for the analysis of the numerical studies. For the current study Multiphase surface tension enabled volume of fluid, RNG k-epsilon model is employed for the analysis. The flow is governed by continuity and momentum equations with turbulence and multiphase (evaporation-condensation) modelling. Flow, volume fraction and turbulence equations are solved for the study. Following assumptions are made in the model:

- Water through micotube rises only due to capillary rise. Inter molecular attraction is neglected.
- Evaporation takes place due to heat flux only.

Surface tension between water and air is chosen as 0.068722 N/m at 30˚C atmospheric temperature at which simulation has been conducted. Heat flux assigned as energy source is 240watts/m^2.

4.1 CAPILLARY RISE DYNAMICS STUDY

Analysis of capillary rise is always difficult when the internal radius of capillary tubes is of microdimensions. Zhmud et al. (2000)[1] experimentally measured the capillary rise dynamics as the depth versus time. This experiment data is verified by means of numerical simulation using ANSYS Fluent. The conservation of mass, momentum is used for modelling the same. The fluid properties used for the study are given below.

- Fluid: Dodecane
- Surface tension, ϒ= 0.025 N/m.
- Density, ρ= 750 kg/m^3.
- Viscosity, η= 1.7x10^{-3}Pa s.
- Angle of capillary, θ= 17˚.
- Radius of capillary tube, r=10^{-4}m.

The steps involved in arriving the plot of head rise versus time was geometric creation, meshing, setting up of solution in Fluid Flow (Fluent), solution and results.

4.1.1 Geometry Creation and Mesh Generation

The simulation of capillary rise is performed on Fluid Flow (Fluent) on ANSYS Workbench. Inside the Fluid Flow (Fluent), geometry was created on Design Modeller and mesh generation was performed in ANSYS Meshing. The steps performed for geometry creation and mesh generation are summarized below,

- In the Design Modeller, the geometry was created with breadth=5000µm and height=6000µm. Now the microtube of diameter 200µm was created. Fig. 1 shows the drawn geometry.

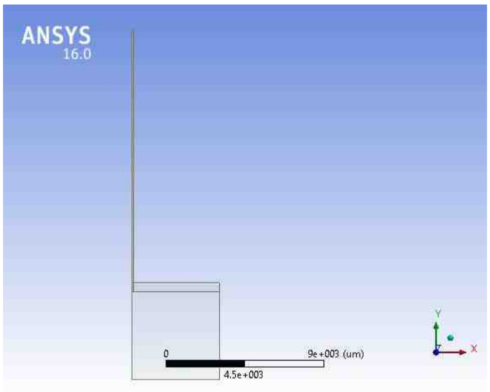

Figure 1-Geometry in the Design Modeller

After drawing geometry meshing is to be done for the analysis. In the Meshing stage of ANSYS Workbench, with an edge sizing of 10^{-4}m and a relevance of 50 it is generated hexagonal elements for the tube portion. By default the entire domain is meshed. The total number of elements obtained after meshing is 112771. The meshed geometry is shown in Fig. 2.

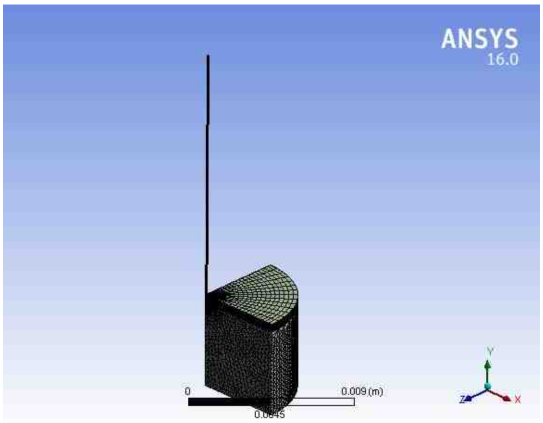

Figure 2-Meshed Geometry

4.1.2 Solution Setup

In Fluid flow (Fluent) inside the ANSYS Workbench, Double precision serial processing and 3D options are selected.

The volume statistics is checked to ensure that it is positive and also checked the mesh quality. The minimum orthogonal quality obtained is 0.110951 (range between 0 and 1), maximum ortho skew is 0.75839 (range between 0 and 1) and maximum aspect ratio is 2.14065(range between 0 and 10). Since the values are appeared in the prescribed range the quality of the created mesh is good.

Using pressure based solver, the velocity formulation was set as absolute. The velocity formulation will result in most of the flow domain having the smallest velocities in that frame, thereby reducing the numerical diffusion in the solution and leading to a more accurate solution. The absolute velocity formulation is preferred in applications where the flow in most of the domain is not rotating. The relative velocity formulation is appropriate when most of the fluid in the domain is rotating. The flow is specified as Transient. Since the flow takes place through the tube to the top in the upward direction (ie, along Y axis) the value of gravitational acceleration is assigned as -9.81m/s^2.

Viscous Laminar, k-epsilon (2eqn) model is selected for the transient analysis from the available viscous models.

Since there is no liquid named dodecane to load from fluent database materials, it has been done by copying some other liquid and changing the properties of corresponding liquid. Note that chemical formula of the new liquid should also be changed (named Dodecane with chemical formula $CH_3(CH_2)10CH_3$) and in the properties the density and viscosity are changed to 750kg/m^3 and 1.7x10^{-3} Pa-s.

Since there are two phases involved in the process the liquid dodecane and air, two phases are assigned accordingly, primary phase as air and secondary phase as dodecane. Interaction between two assigned phases is to be set and hence surface tension force modelling is selected and activated wall-adhesion. The surface tension coefficient between dodecane and air is assigned as 0.025N/m.

Now the boundary conditions are set as walltube, walltank, wall air, pressure inlet air, outlet air, symmetry1 and symmetry2. For the walltube boundary condition the contact angle is assigned as 17˚. For walltank there should not be any capillary rise and hence it is assigned as 90˚.

A region of water adapted for a patching height of 5mm, formed a cylinder of water region using surface monitors in the solution setup, monitor is set for plotting the head rise versus time plot. The head rise is plotted by setting a plane in the tube in line with the top level of water in the tank. The plane is set at a height of 5mm from bottom of the tank in the tube for getting the head rise at each time step. The same can be done by autosaving case and data files. For transient analysis, calculations are done for each time step. After setting the monitors, solution initialization is done using Standard Initialization.

The simulation is carried out and the graph of head rise versus time was plotted. Fig. 4 shows the comparison of plot of head rise versus time graph obtained from CFD simulation and experimental study from journal paper. In the paper they compared the head rise plot with that obtained from the Lucas-Washburn Equation.

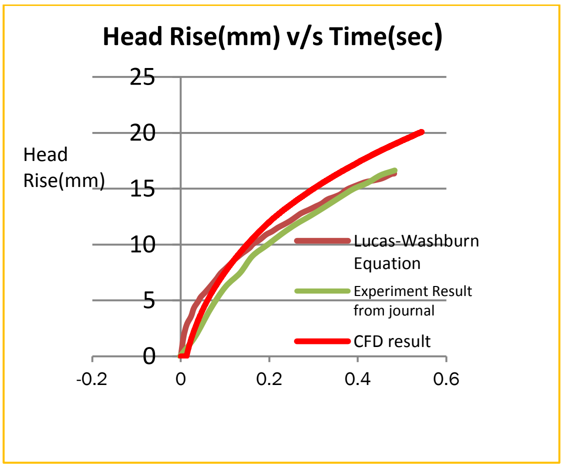

Figure 4-Comparison of plot from CFD result with the experimental result

From the plot obtained it can be inferred that the CFD result shows good agreement with the experimental result but with slight deviation in magnitude. The capillary rise observed shows that the use of CFD for the analysis can yield similar result with the experiment. Hence modelling using CFD for capillary rise is working properly and hence the methodology is considered as validated.

4.2 FLOW THROUGH MICROTUBE AND EVAPORATION RATE

Microtube is of prime importance in many field applications like medical field, civil engineering field etc. If there is no pressure induced to assist the flow through microtubes the flow may arise due to capillary effect under such field conditions. For the tubes with top end exposed to atmosphere the evaporation follows capillary rise under normal atmospheric conditions. The analysis of flow through xylem channels which are of microdimensions in trees has been conducted by researchers using isotopic tracers. Apart from the experimental study the numerical study takes the benefit of low cost of study and accuracy. The study of capillary rise in microtubes followed by evaporation is carried out using CFD simulation in ANSYS Fluent.

The steps involved in Fluent are geometry creation, mesh generation, set up, solution and results. The CFD simulation of evaporation from a microtube is conducted with same dimensions and conditions of the experimental setup made for the validation of the result. The tube with same dimension as that of experimental setup is created in CFD for capillary rise study followed by evaporation rate analysis. The steps involved in Fluent are geometry creation, mesh generation, set up, solution and results. Capillary rise is assisted by surface tension force and evaporation is assisted by heat flux.

4.2.1 Geometry Creation and Mesh Generation

Inside the Fluid Flow (Fluent), the geometry is created on Design Modeller and mesh generation is performed in Ansys meshing. Since the flow takes place only in a single direction (ie, upward direction) and also due to symmetry of the geometry the problem can be studied by 2d simulation. The dimensions of microtube and entire domain selected for the study is given in table 1 below.

Table 1- Details of dimension of microtube

Geometry	Length	Height/dia
Outer domain	7 cm	6cm
Microtube	3.5 cm	500 µm

The steps performed for geometry creation and mesh generation are summarized below.

- In the Design Modeller, a rectangle for the outer domain of air with length=7cm mm and height=6cm and the tube length=3.5cm and breadth=0.5mm are drawn.
- Surfaces are generated corresponding to each sketch.
- Used Boolean option to substract the body of tube from the outer domain. Fig. 5 shows the created geometry.

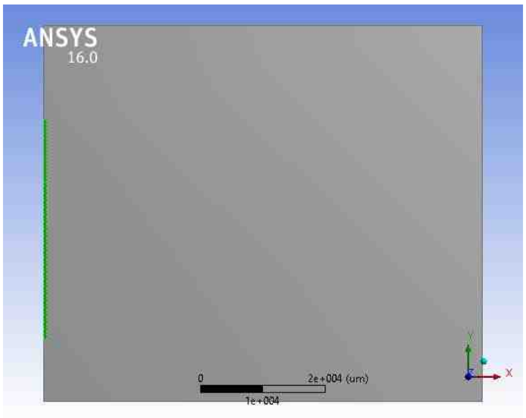

Figure 5-Geometry of Microtube

In the Meshing stage of ANSYS Workbench, with an edge sizing of $5x10^{-4}$m for the tube and a relevance of 100, it has generated and obtained tetrahedral elements for the tube portion. The total number of elements obtained from meshing is 9587. The meshed geometry is shown in Fig. 6

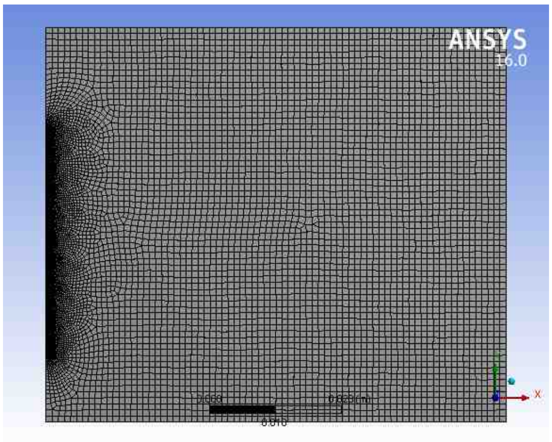

Figure 6-Meshed Geometry

4.2.2 Solution Setup

In Fluid flow (Fluent) inside the ANSYS Workbench, double precision option, serial processing option and 2D options are selected. The quality of meshed geometry is checked and reported. Volume statistics checked to ensure that it is positive. In mesh quality, the minimum orthogonal quality is 0.77547(range between 0 and 1), maximum ortho skew is 0.224526 (range between 0 and 1) and the aspect ratio is 3.03127. Since the values are appearing in the prescribed range, the mesh quality is good. The various options assigned for the analysis is described below.

- Pressure based solver, relative velocity formulation and time as Transient are set. Since the flow takes place from fluid through the tube to the top in the upward direction (ie, along Y axis) the value of gravitational acceleration is assigned as -9.81m/s^2. Axisymmetric condition is set for studying the problem since symmetry exists.
- Volume of fluid model selected with number of Eulerian phases increased to 3 (viz, air, water, vapour are included in the domain). Energy kept on since evaporation to be started due to the heat flux applied at the top. Viscous model with k-epsilon (2eqn) model is selected from the available viscous models.

- Two materials water-liquid and water-vapour are loaded from the fluent database materials and copied.

- 3 phases are defined since there are 3 phases included in the process, water, vapour, and air. Setting air as primary phase secondary phases phase-2 and phase 3 are set as Water–Liquid and water vapour respectively. The phase interaction between water-liquid and air is assigned as 0.068722 N/m.
- Operating temperature 303(k) and operating density chosen is 0.5542kg/m^3.
- Since evaporation taking place from the top of the tube the mass transfer mechanism is increased to 1 and it is chosen as Evaporation- Condensation mechanism. Number of energy sources is increased to 1, its value is assigned as heat flux of 240watts/m^2.

- The utilized boundary conditions are walltube, walltank, symmetry and outlet. For walltube the contact angle between water and air is taken as 0˚. The contact angle between all other phases is set as 90˚. The surface contact angle between all phases is assigned as 90˚, since there is no capillary rise assumed along the walltank boundary. Wall is assigned as stationary wall with no slip condition. A region of 3.5 cm height of water forming a quad is adapted to patch the outer domain of air with water upto a height of 3.5cm. For transient analysis, calculations are done for each time step.

- After the solution initialization water is patched upto the adapted region in the previous step specified. Using surface monitor the mass flow rate versus time graph is plotted. In the Run Calculation step, 0.001 is given as the time step size and 30000 is given as the number of time steps for getting simulation of the result for 30 seconds. The plot obtained as mass flow rate versus time graph is shown below in Fig. 7.

Animations are set for visualizing the phenomenon of capillary rise followed by evaporation.

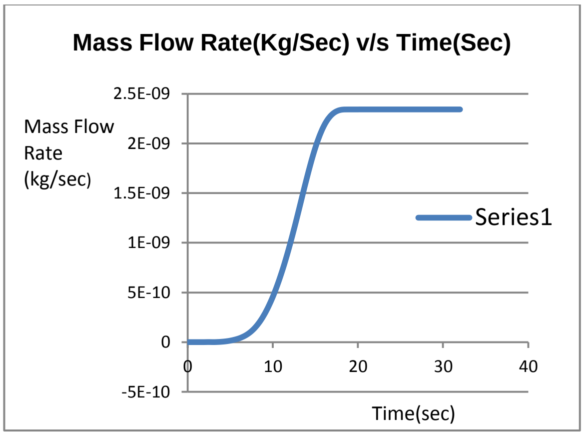

Figure 7-Mass flow rate versus time graph

5.0 EXPERIMENTAL STUDY

The experimental study setup consist of the following materials,

- A container with weighed quantity of water
- A plastic membrane to cover the top of water surface
- A microtube of 1mm diameter and 3.5 cm height

Period of duration for the study is set for 24 hours, the atmospheric temperature was 32̊C.

6.0 RESULTS AND DISCUSSIONS

Capillary flow through a tube is analyzed first and validated with the published experimental data entitled "Dynamics of Capillary Rise" by Zhmud B. et al. (2000)[1]. Special attention is paid to the capillary rise of surfactant solutions. The head rise versus time graph plotted from the results obtained from CFD simulation with the properties of fluid as chosen in the literature. From the plot obtained for the capillary rise from CFD, it is clear that the capillary rise dynamics can be well represented by CFD simulation. The results show that the numerical modeling attains better agreement with the literature results.

The phenomenon of capillary rise followed by evaporation is studied by considering the flow in a microtube. The diameter of tube chosen for the study was of 0.5mm. Capillary rise is noticed in the geometry for a simulation lasting for 30 seconds. The operating temperature was assigned as 303K. From the mass flow rate versus time graph obtained from CFD simulation, it can be inferred that the evaporation notably started from 4.12^{th} sec and the evaporation rate get stabilised at 18.24^{th} sec. The maximum evaporation rate once the rate has stabilised is found to be 0.2021g/day (i.e., $2.34X10^{-9}$ kg/sec). From the experiment the result obtained was 0.39 g/day, and the variation noticed is due to the little higher atmospheric temperature when the experiment had been conducted (put under the direct sunlight). CFD results are comparable with the experimental results.

7.0 CONCLUSION

The flow simulation of capillary rise has been carried out and found to be matching with experimental result but with slight deviation in the magnitude. Capillary rise followed by evaporation in a microtube is modelled using CFD package ANSYS FLUENT and found that simulation results are comparable with the experimental results.

REFERENCES

[1] Zhmud B., Tiberg F. and Hallstensson K. 2000, **"**Dynamics of Capillary Rise"**,** *Journal of Colloid and Interface Science,* vol. 228(2); pp. 263 – 269.

[2] Richards L. A. 1931, "Capillary conduction in Porous Mediums", *Review Article, Google.*

[3] Lu N. and Likos W. J. A. 2004, "Rate of Capillary Rise in Soil", *Journal of Geotechnical and Geoenvironmental Engineering.*

[4] Cazadero 2004, "Trees and Capillaries", *Review Article-Google*

[5] Vasu K. and Wahid P. A. 1990, "Measurement of Transpiration Rate in Coconut Palm with Tritiated Water: Tritium Profile in Coconut Crown", *Journal of Nuclear Agriculture and Biology, vol 17: pp. 110-112.*

[6] Zhang X., Coupland P., Fletcher P. D. I. and Haswell S. J. 2009, "Monitoring of liquid flow through microtubes using a micropressure sensor" *Chemical Engineering Research and Design- Elsevier.*

[7] Rawool S. A., Mitra S. K. and Kandlikar S. G. 2006, "Numerical Simulation of Flow Through Microchannels with Designed Roughness", *Microfluid Nanofluid,* vol 2, pp. 215–221.

[8] Lu N. and Likos W. J. A. 2004, "Rate of Capillary Rise in Soil"*, Journal Of Geotechnical And Geoenvironmental Engineering.*

[9] Neema T. 2006, "Determination of the Transpiration Rate of the Different plant Species within the Namibrand Nature Reserve Area", *Thesis Report.*

[10] Jie J. R., Quran S. F. and Mei H. 2006, "Flow Characteristics of Deionized Water in Microtubes", *Proceeding Letter, Chinese Physics Letters,* vol 23 (12).

Presenting author Biodata

Name : JOUHARA M. K.

Designation : Student

Company : Govt. Engineering College, Thrissur,
Thrissur – 680009

E-mail : jouhararahman6@gmail.com

Mob : 9633920980

Qualification : Pursuing M. Tech

Area of Expertise **:** Water Resources and Hydroinformatics

Significant Achievements **:** NA

Number of Papers Published in Journals **:** NA

Number of Papers Published in Conferences: NA