The Iraqi Journal For Mechanical And Material Engineering, Vol.10, No.1, 2010

Fluid-Particle Flow in Deeming of Inertia-Air Filter Design

By

Khalid Salem Shibib
Assistant Prof.
Department of Laser Department of Optoelectronics Engineering

Karema Assy Hammad
Assistant lecturer
Electro-mechanical Engineering

University of Technology-Baghdad

Abstract:

The usage of inertia air filter to clean air passing to air consuming device has few attention in spite of its importance .This work focuses on the idea and advantages beyond this filter .A finite element solution of Navier- stoke equation in turbulent Zone is used to obtain pressure and velocity distribution through typical section in inertia air filter, these values are used to obtained the trajectory of different particle size using four order Runge-kutta integration method .The path trajectory will device on how large particles size can enter certain zone. Inertia air filter will prevent the air consuming devices from stopping due to blocked air filter .A special design is proposed so that to minimize loss with good ability in filtering dusty air .The usage of inertia air filter will help in elongate the life time of the usual air filter without appreciable increasing in pressure loss.

جريان المائع و الجسيم في تصميم فلترالهواء العامل بمبدأ القصور الذاتي

خالد سالم شبيب قسم هندسة الليزر والبصريات الاليكترونية -الجامعة التكنولوجية

كريمة عاصي حمد قسم الكهروميكانيك- الجامعة التكنولوجية

مختصر

أن استخدام فلاتر الهواء المعتمدة على مبدأ القصور الذاتي لتنقية الهواء لأجهزة استهلاك الهواء لم تلقى اهتماما واسعا رغم أهميتها .هذا البحث يركز على الفكرة والفوائد المستنبطة من هكذا فلاتر. استخدمت طريقة العناصر المحددة لحل معدلات Navier-stoke في الجريان المضطرب لاستخراج قيم الضغط والسرعة خلال مقطع نموذجي لفلتر يعمل بمبدأ القصور الذاتي والتي ستستخدم لاحقا لإيجاد مسار أحجام مختلفة من الجسيمات باستخدام طريقة Runge-kutta للتكامل وبالاعتماد على هذا المسار سيتم معرفة حجم الجسيمات الذي سيدخل منطقة معينة .هذا الفلتر سيستخدم كمانع لانسداد الفلاتر العادية والذي يودى إلى إيقاف الأجهزة المستهلكة للهواء. .تم اقتراح تصميم خاص لتقليل الخسائر مع قابلية جيدة لتنقية الهواء المغبر.أن استخدام الفلاتر العاملة بمبدأ القصور الذاتى سيطيل عمر الفلاتر العادية بدون زيادة مؤثرة بخسائر الضغط.

1. Introduction:

Many authors established the criteria of solving fluid- particle flow that is to study the behavior of particle flow by virtue of their carrying fluid [1,2,3].Its application have many important feathers such as studying behavior of fuel particle during its injection in internal or external combustion engine , also the application of this method is in studying paths of the particle during multi-phase flow .In this work ,its benefit can be extended to study the air filtration so that air with contaminated particle (such as dust) can be cleaned to some extent so that to protect air handling units , diesel engine and stationary gas turbine from the harmful effect of these particles.

This proposed particle separation method is very convent to frequently dust weather where it can be an assistance air cleaner so as to increase the life of usual air filter .From practical experience ,it is known that the dust storm may blocked the filters and the air handling units can no longer maintain their duty without changing or cleaning the usual air filter .By using this new filter design one can insure that no harmful dust will flow to the usual filters in air-handling unit or other clean air consuming devices .Using bleed duct system incorporated in the inertia maze design ,collected dirt particles may be ducted to any position for disposal so that large particle can be purified from air and the other small particle can be cleaned by usual air filter.

The inertia air separator is designed to handle large volume of heavily contaminated air within an extremely confined space without appreciably increasing pressure drop.

The inertia maze separator is essentially vee-shaped, the sides consisting of a series of identical equally spaced vanes providing narrow passages through which the clean air can flow .The apex of the vee is open to a bleed slot at the rear of the cell, which is in turn connected to a common bleed duct connecting all bleed slots of a particular assembly see Fig 1. The sharp turns through the narrow side openings the major passes to the clean side of the filter, whilst dirt particles because of their greater mass tend to continue on a straight path to be exhausted via the bleed slots by 10% bleed air flow. The actual air flow through the inertia maze separator is normally 100% of the desired clean air flow, 90% being drawn by the clean air consuming device. High velocity in the secondary air bleed duct prevents any dirt build-up.

Due to high cost of air filter used in air handling units, one think of how one can increase its running life in frequent dust storm weather, the solution is by using the inertia air filter.

Theoretical and experimental investigation for inertia air filter design is introduced where the aim is to ensure good characteristic of air filtering in frequent desert storm weather.

2. Theory:

A suitable section is chosen to formulate the problem mathematically see fig 2, where the continuity, momentum has to be satisfied

2.1 Design of inertia air filter

The proposed design of inertia separator is shown in fig 1.with 24 vee-shaped in each side the angle of the apex of adjusting fans is of order of 6 degree .A distance of 2 cm between each apex of vee-shaped .The inlet section has a dimension of 0.126x 0.5 m design .The depth of the filter is 0.5 m see fig1. The typical section which can formulate the physical domain is shown in fig2.The proposed material is an ant-erosion material (such as stainless steel) in order to resist the erosion due to high speed of particle size that may strike the surface of the filter material .

2.2 Partial differential Equation

2.2.1. Navier-Stoke equation

The flow is assumed to be unaffected by the particle flow where the velocity and pressure profile through the shape is the aim ,the finite element formulation of NSE is established well in many references using primitive variable method with four-node element and equal-order shape function for pressure and velocity , the model of turbulence is the log wall model near the wall while a k-ε model is used for the turbulent core .The coupled equations governing the fluid flow are solved using SIMPLER(Semi Implicit Method for Pressure Linked Equations Revised) algorithm [4] with total number of element of 588 that contains 667 node distributed as shown in fig 2 .Concentration of nodes near expected large gradient of velocity and pressure are favorable to insure good accuracy . The algorithm, originally developed for finite difference method, has been modified and proposed by several researchers for finite element method[5,6].A widely used technique known as streamline up-winding method is used with four quadrilateral element, where the convection terms is evaluated directly along the local stream lines [7].

2.2.2 Particle trajectory

From the fluid velocity, particle motion is computed as fallow:

Neglecting other forces influenced the particle, equation of motion can be written as[2]:

$$\sum F_i = m_p\, a_i$$

but $\sum F_i = F_d$ and $F_d = -\rho_f C_d A_p v_{r,i}^2 / 2 = \rho_p a_{p,i} V_p$

(1)

Where

$$A_p = \frac{\pi d_p^2}{4} \quad , V_p = \frac{1}{6}\pi d_p^3 ,\ m_p = \rho_p V_p$$

Then

$$a_{p,i} = \frac{dv_{p,i}}{dt} = -\frac{3\,\rho_f C_d}{4\,\rho_p d_p} \left| v_r \right| v_{r,i}$$

(2)

where the $v_{p,i}$ is the i component of the particle velocity and $v_{r,i}$ is the particle relative velocity[2]:

$$v_{r,i} = v_{p,i} - v_{f,i}$$

(3)

$$C_D = \frac{24}{\mathrm{Re}_p}(1 + 0.15\,\mathrm{Re}^{0.687})$$

(4)

$$\mathrm{Re} = \frac{\left| v_r \right| d_p \rho_f}{\mu_f}$$

(5)

Equation of motion (1) is appropriate for dilute gas-solid flow in which the particle size is small compared to the smallest length scales in the carrier flow and for which other forces acting on the particle (added mass, history force, etc.) are substantially smaller compared to the drag owing to the large density ratio. [8]

The process after solving NSE to find the trajectory of the particle is as follow:

find the new proposed location of the particle from the initial known particle velocity (it is assumed at entrance that the velocity of fluid is equal to the velocity of particle (i.e. zero relative particle velocity) and hence find the instantaneous velocity of fluid at new location from which find the new velocity of the particle an iteration solution is required to average the velocity of fluid with the two point then the solution is repeated to insure good accuracy and so on ,the solution is tested for different particle diameter from 1 μm to 20 μm and the density of dust is assumed to be 1520 kg/m^3, the density of air is taken for standard atmosphere .

In this work, the fluid flow is not affected momentum exchange with the particles and the (undisturbed) fluid velocity required in (1) is the value interpolated to the particle position that is computed in the Large Eddy Scale solution.

It important to know that the solution is important only in certain paths where there is a possibility of certain particle diameter to follow the fluid velocity so that to enter a region where the air must be clean, so only these paths will be studied .The process of obtained particle trajectory can be divided into three steps

A. location

To determined the location of a particle in finite element mesh ,a test must be made for the nearest node to the new location from which the four elements containing this node must be tested to know which one is containing this node ,then the element containing this location is known by comparing the max. and min. coordinates of each element.

B .interpolation

By knowing the element that contains the new location an interpolation process has to be carried out to determine the component of velocity (i.e. vx and vy) and pressure if desired. Some useful four-linear interpolation may carry out assume only four field variable at each interpolation process is known.

C. integration

Given the solution at the end of n time steps U_p^n, X_p^n for particle and U_f^n, X_f^n for fluid then compute U_p^{n+1}, X_p^{n+1} see [9], then

For I=1 to K

$$U_p^{*n,I} = U_p^{n,I-1} + \frac{\Delta t}{K} F_1$$

where $F_1 = -\frac{3}{4} C_d \frac{\rho_f}{\rho_p d_p} (U_f^n - U_p^{*n,I})$ (at $X_p^{n,I-1}$)

$$X_p^{*n,I} = X_p^{n,I-1} + \frac{\Delta t}{2K} (U_p^{n,I-1} + U_p^{*n,I})$$

$$U_p^{n,I} = U_p^{n,I-1} + \frac{\Delta t}{2K}(F_1 + F_2)$$

where $F_2 = -\frac{3}{4} C_d \frac{\rho_f}{\rho_o d_p}(U_f^n - U_p^{*n,I})$ (at $X_p^{*n,I}$)

$$X_p^{n,I} = X_p^{n,I-1} + \frac{\Delta t}{2K}(U_p^{n,I-1} + U_p^{n,I})$$

next

then $U_p^{I+1} = U_p^{n,K}, X_p^{n+1} = X^{n,K}$

A special model is used to formulate the path of particle that strikes the surface ,when certain particle hits the surface an inverse velocity is imposed depend on the direction of the particle if the direction of the particle is perpendicular to the surface then both component of velocity will be inversed and if the direction of path of the particle makes an angle greater than that of the perpendicular to the surface then only one component of the velocity will be inverse depends on the direction of the surface.

2.3. Boundary conditions

The entering velocity will be tested with 10 and 15 m/sec with no relative velocity at the entrance of the filter .Special attention has to be paid for imposing boundary conditions since continuity must be satisfied (see fig. 2) also one outlet of zero pressure may be applied to obtained pressure distribution through the section of filter. While zero velocities are applied on solid surfaces, the inlet velocity may be either 10 or 15 m/sec, also the boundary vertical velocity at the far side may also be zero (i.e. far away from boundary layer).

3. Results and Discussion

The solution of Navier stock equation lead to obtained velocity and pressure distribution through the section of inertia filter for inlet velocity of 10 and 15 m/sec. The velocity distribution is shown in fig 3 for inlet velocity of 10 m/sec. Also the pressure distribution for this velocity distribution is shown in fig 6. The trajectory and the inlet position for different path and size of particle are shown in fig 4.

For inlet velocity of 10 m/sec the maximum particle that allowed to pass to the region of intended clear air zone is less than 18 μm ,since particle of that size will strike the surface of the vee-shaped vans and go back to a region of turbulent core where it will fallow the path of other particle see fig 4 ,this is for upper path ,the next path show that only particle less than 3 μm will enter the clean Zone, while the particle with diameter> 3 μm will go throw the bleeding stream fallow the main air stream. Its worth while to say that any particle with that size, down this path (i.e. diameter $\geq 3\mu m$ will also go with the bleeding air toward the exit by virtue of its inertia even particles has less diameter .The pressure loss across two vee-shaped fan is found to be 67.32 Pa theoretically and by using empirical formula of pressure loss across duct its found to be 64.22 Pa and via bleeding duct 1.1 Pa .

Increasing inlet velocity 15 m/sec will lead to decrease in size of particles that are allowed to inter the region of intended clean zone (the upper path in fig 5) where the particle seems to follow the air enter via vee- shaped fans is 13 μm and less while for the next path (the lower one) the particle size allowed to pass through the vee-shaped fans is reduced to 1 μm while a particle of 2 μm diameter will strike the wall and go toward the exit .The friction loss across two vee-shaped fans is found to be 143.64 Pa, theoretically and by using empirical formula for pressure loss of ducting, it is found to be 138.14 Pa and via bleeding duct 2.04 Pa.

Experimentally ,The dust is gathered after a dusty storm which its diameter is found in a rang of 200-0.5 μ m and injected by gravity to the tested section which combined from the two symmetrical section shown in fig.2 ,the collected particle for velocity of 15 m/s is verified the result where there is no particle larger 13 μ m is found in the air that passes to the intended clean zone assume the particle is gathered be a very fine air filter and tested by microscope .

4. Conclosion

Without high increase in pressure loss an efficient inertia air filter is designed .Some new idea is introduced to clean air carrying dust particle even the theory is well understood .Intended clean zone with less cost is the aim , even the usual air filter doing their duty well but they will blocked after heavy dust storm .By using some shape of inertia filter ,long life with full done duty will be expected for usual air filter , this will be insured by some initial cost to have an inertia filter and some additional maintenance on the bleeding air fan .Some ideas may be present on how one can increase the attraction force on the particles so as to force them to go through certain path .Increasing inlet velocity may be one of them but it has some limitation since pressure loss will increased as velocity increases, the other method is by using an additional electronic air filter fixed with this filter so as to increase the whole efficiency .

References:

1) Hersir Sigurgeirsson,(2001) Particle Field Models: Algotartim and Application, Stanford University, PhD thesis.

2) Marion W.Vance (et.al)(2004)" Properties of the Particle Velocity Field in Gas-Solid turbulent Channel Flow" 5th International Conference on Multiphase Flow, Yokohama, Japan.

3) Denis Blackmore (et.al), (1999), New Mathematical Models for Particle Flow Dynamics, Journal of Nonlinear Mathematical Physics 6(2), 198–202

4) Patankar SV (1980) "Numerical Heat Transfer and Fluid flow", Hemisphere, Washington DC.

5) Prakash C, Patankar SV (1985)," A control volume-based F.E.M for solving the N.S.E Using Equal order velocity pressure Interpolation. Numerical Heat transfer 8(1), 259-280.

6) Haroutunian V, Engelman MS and Hasbani I(1993)," Segregated Finite element Algorithms for the numerical solution of large scale compressible Flow problems". International Journal for numerical Method in Fluid 17(1), 323-348

7) Rice JG and Schnipke RJ. (1985)," A Monotone Streamline Upwind F.E.M for Convection-Dominated Flow", Computer method in Applied Mechanics And Engineering 48(1),313-327

8) R.Avila and M.A.RODRIGES-Meza, (2004),"Velocity auto correlation function of a dispersion of heavy particles in a turbulent flow", Revista Mexicana de Fisica 50(2), 156-161

9) Chun-Yen Chow, (1979),"An Introduction to Computational Fluid Mechanics", John Wiley and Sons, Inc, Washington, USA.

Nomenclatures:

A- area (m2)

a - acceleration (m/sec2)

C- drag coefficient (dimensionless)

d –diameter (m)

F -force (N)

U -velocity vector (m/sec)

Re –Reynolds number

X -displacement (m)

t -time (sec)

v -velocity (m/sec)

V-volume (m3)

K-no of loops

Greek

ρ -density (kg/m3)

μ -viscosity (Pascal .sec)

Subscript and superscript:

o-initial

d-drag

r -relative

i- vector component

P- Particle

f-fluid

*- iterative solution

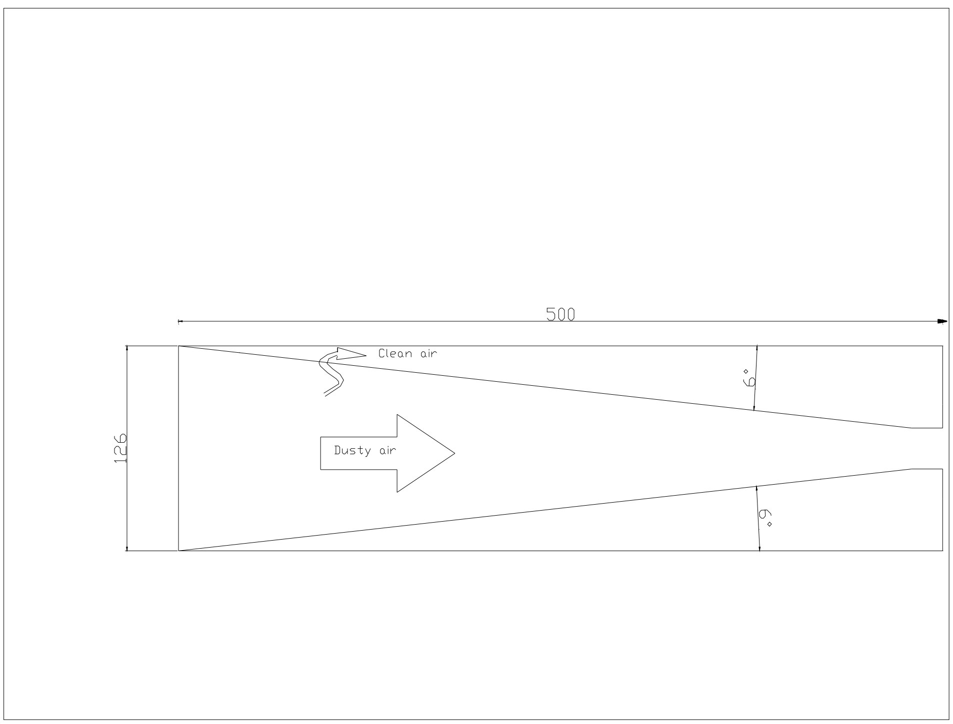

Figure 1. Inertia air filter -Basic dimension in mm.

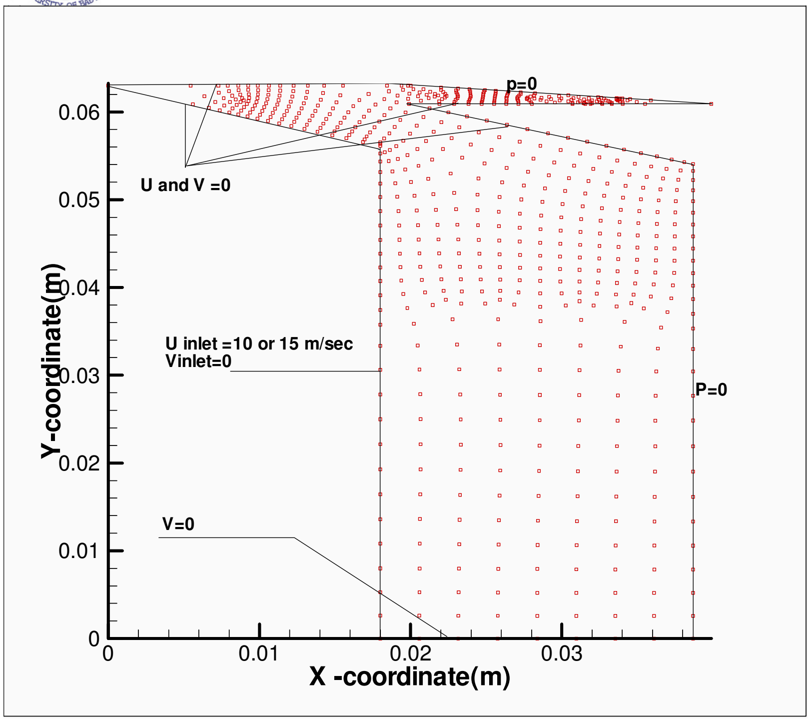

Figure 2. Indicates the part of the shape that has been studied to determine the fluid and particle flow.

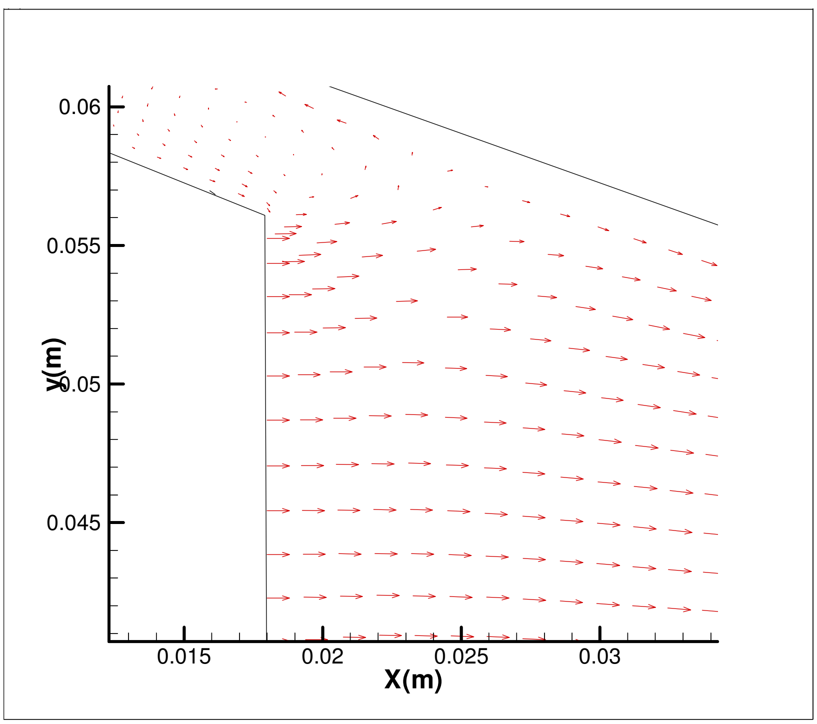

Figure 3. Velocity distribution through the upper part of Fig 2.

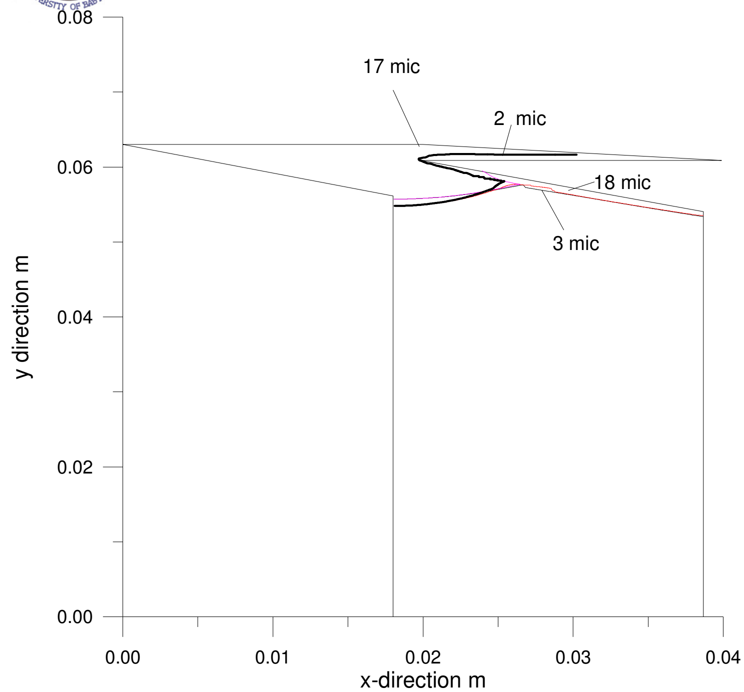

Figure 4.Path of different particle size for different inlet location at inlet velocity of 10 m/s

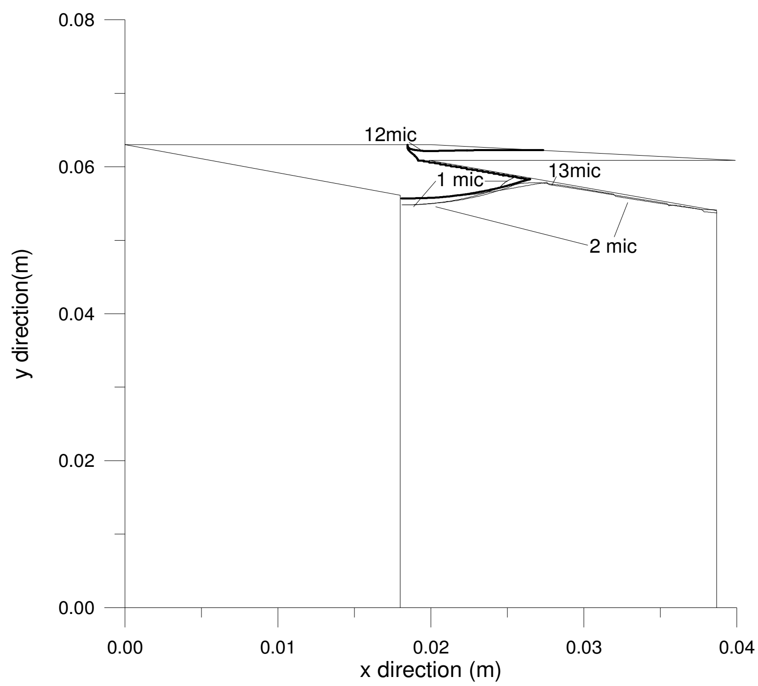

Figure 5 Path of different particle size for different inlet location at inlet velocity of 15 m/s

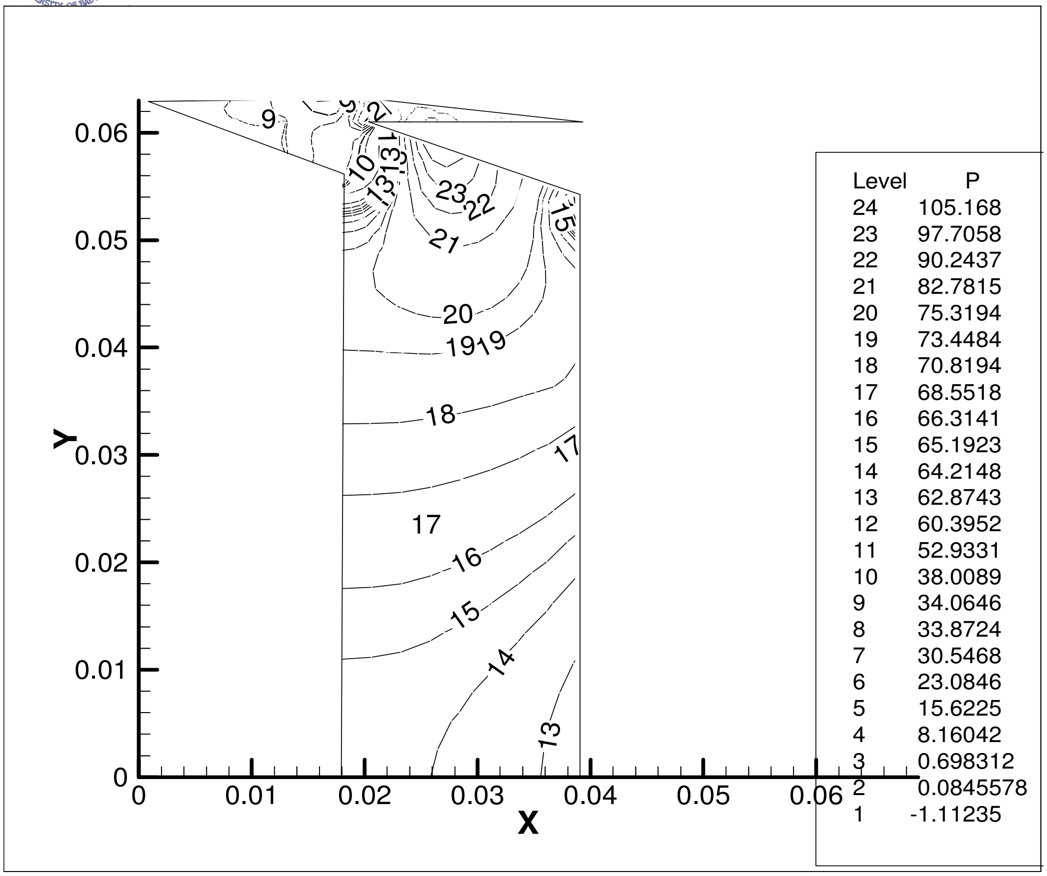

Figure 6. Pressure distribution for inlet velocity of 10m/sec