International Journal of Engineering, Mathematical and Physical Sciences
ISSN: 2517-9934
Vol:5, No:6, 2011

Use of Time-Depend Effects for Mixing and Separation of the Two-Phase Flows

N. B. Fedosenko, A.A Iatcenko, S.A. Levanov

***Abstract*—**The paper shows some ability to manage two-phase flows arising from the use of unsteady effects. In one case, we consider the condition of fragmentation of the interface between the two components leads to the intensification of mixing. The problem is solved when the temporal and linear scale are small for the appearance of the developed mixing layer. Showing that exist such conditions for unsteady flow velocity at the surface of the channel, which will lead to the creation and fragmentation of vortices at Re numbers of order unity. Also showing that the Re is not a criterion of similarity for this type of flows, but we can introduce a criterion that depends on both the Re, and the frequency splitting of the vortices. It turned out that feature of this situation is that streamlines behave stable, and if we analyze the behavior of the interface between the components it satisfies all the properties of unstable flows. The other problem we consider the behavior of solid impurities in the extensive system of channels. Simulated unsteady periodic flow modeled breaths. Consider the behavior of the particles along the trajectories. It is shown that, depending on the mass and diameter of the particles, they can be collected in a caustic on the channel walls, stop in a certain place or fly back. Of interest is the distribution of particle velocity in frequency. It turned out that by choosing a behavior of the velocity field of the carrier gas can affect the trajectory of individual particles including force them to fly back.

***Keywords*—**Two-phase, mixing, separating, flow control

I. INTENSIFICATION MIXING IN MICROCHANNALS

IN typical microfluidic applications Reynolds number Re is low due to small dimensions. Because of it, very effective process of turbulent mixing is not arises and cannot be used in these scales. In this situation processes of diffusion and convection have not enough space and time for effective mixing. In such a case, fruitful idea to use the chaotic advection instead of turbulence mixing looks very attractive.

Concept of chaotic advection was suggested in 80th but, nowadays, due to microfluidic applications it obtains a new vision and possibilities for development. It is proved that a necessary condition for generation of chaos is the crossing of streamlines at two consecutive time instants [1]. Mathematical description of this idea is well known as “linked twist maps concepts” [2]. Furthermore, many experimental results also confirm this fact.

But, this theory has one disadvantage: it may be used for analysis of mixing quality only “a posteriori”. Theory investigates conditions of the inner flow and can answer the question - became chaos or not. It can be used to analyze quality of the mixing and compare two different devices. But, it is useless in design of microfluidic devices because at the beginning of design we know nothing about inner flow and we only want to create desired flow.

Expressed earlier necessary condition for chaos generation will not work if we “place” it in the boundary of the domain instead of the inner flow. The main idea of this work is to devide procedure in two steps. First, we create necessary condition at the boundary. Then, we “blow up” it inside flow domain.

II. FORMULATION OF THE PROBLEM

The flow of two fluids – air and nitrogen – inside micro mixer with rectangular constant cross section is under consideration. These gases have very similar physical parameters. It makes their mixing more difficult because of absence of initial gradient of any parameter inside mixer. The scheme of mixer is shown on fig. 1, dimensions are in millimeters. Nitrogen injects into the air through the injector of 1 mm in diameter. Here and then we will base on the determination of advection proclaimed in [1]. According to it, the main idea of this research is attempt to produce conditions in which interface of two gases will have maximally possible length. In so far Re number is small, normal and tangential stresses inside flow have the same order. So, in general it exist two possible ways to act upon fluid flow studied here: to create pressure gradient along channel and to create tangential stress at some cross section which affect upon flow. Evidently, the last way is much more effective at scale under the study. It was realized in few types of micro mixers with moving surfaces [3]. Such moving elements are used in this study, too. Their placement is shown on fig. 1. The aim is to regulate velocities of moving walls in the manner whichh allow to generate the sequence of vortexes inside the flow. They will involve gas interface and, in such a way, to increase it’s length. Finally, it originates conditions for chaotic mixing.

Fig.1 Calculation domain with boundary conditions

N. B. Fedosenko is with the St.Petersburg State Polytechnical University (e-mail: fedosenko_n@mail.ru).
A.A.Iatcenko is with the St.Petersburg State Polytechnical University (e-mail: anuta_sf18@mail.ru).
S.A.Levanov is with the St.Petersburg State Polytechnical University (e-mail:lev-unkind-@rambler.ru).

An instrument of the study is numerical simulation. Flow pattern is described by Navier – Stokes equations, interphase interaction is not taken into account. Uniform air flow (V = 10m/s) puts as initial condition at cross section (X=0). Reynolds number Re = 7.8 is determined with diameter of injector.

Further research is broken on two steps:

i. Study of planar flow (2D)
ii. Generalization of obtained results on 3D flow

During research, required flow pattern is studied in details at initial scale (fig. 1) than scale decreases in 10 and 100 times.

III. TWO – DIMENSIONAL FLOW

The following conditions are realized: p = 105 n/m2, T = 293K, velocity of main air flow and injection of N2 are the same and equal to 10m/s, duration of monitoring is 4 seconds. Wall velocities are varied from 0 till 2m/s. Laws of their regulation are shown on fig. 2 (blue – upper and red – lower walls, respectively).

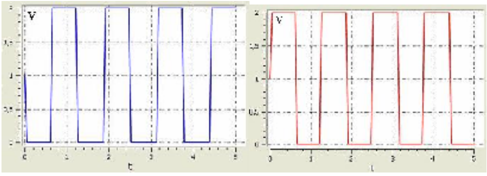


Fig.2 Moving walls velocities

Triangle grid is used with 21444 nodes which concentrate near injector and moving walls. Numerical simulation allows to observe an origination of sequence of vortexes. First vortex originates due to input of shear stress which changes initial profile of velocity. Vortex stretches but does not move downstream. In so far walls moves in contradiction (see. Fig.1), initial vortex brakes into two, three, than four ones. They rotates in same direction. Figure 3 shows development of this process at three moments of time: 0.5, 0.6 and 0.7 seconds (marks 1-3).

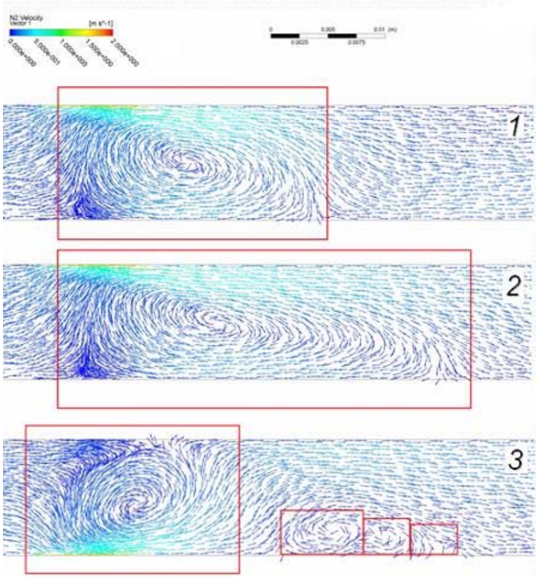


Fig.3 Vortex breakup (time: 0.5, 0.6 and 0.7 seconds)

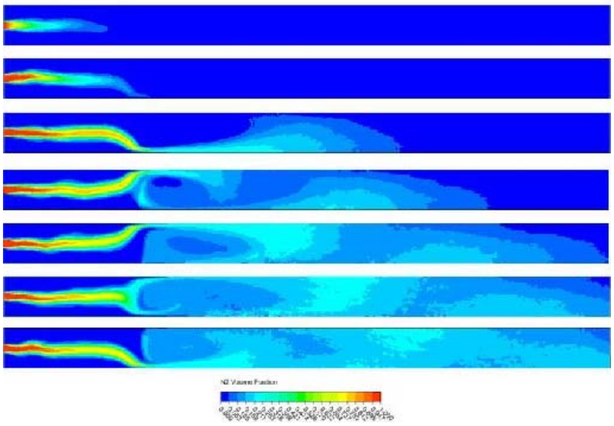


Fig 4 Concentration fields in time from 0.1 till 1.6 seconds

Fields of concentration of injected N2 are shown on fig. 4. It depicts proceeding of mixing due to chaotic advection in time from 0.1 till 1.6 seconds with interval 0.1 sec.

IV. THREE – DIMENSIONAL FLOW

In this case the same law of motion of moving upper and lower walls are used (fig. 2); side walls do not move. Tetrahedral grid is used for simulation with 915694 elements which concentrate near injector and moving walls.

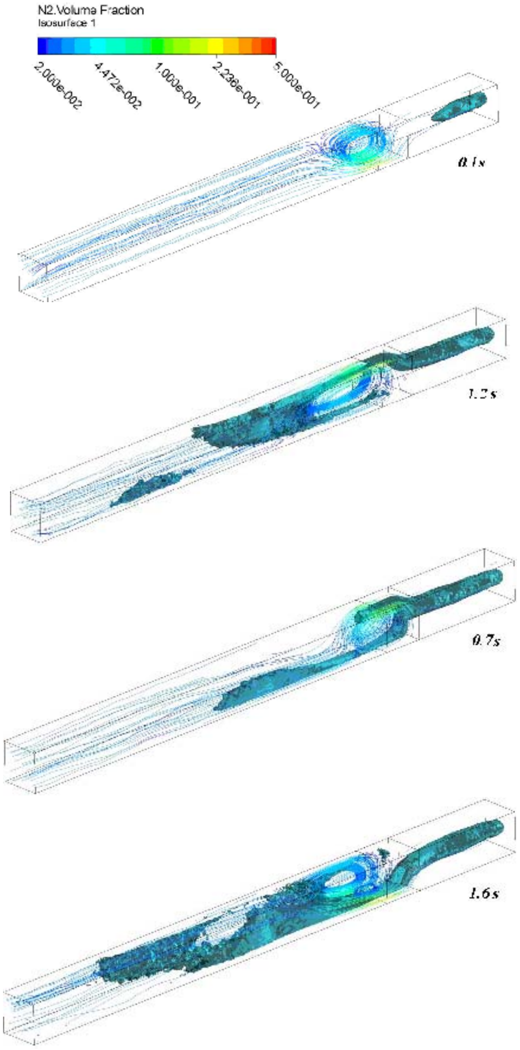


Fig.5 Three dimensional vortex breakups and concentration fields

On fig. 5 instantaneous streamlines of N2 are placed. As in 2D case it is visible the development of mixing with time. It means that suggested method of regulation of wall velocity gives an effective instrument for intensification of chaotic advection.

The following fact has to be mentioned: field of concentration of N2 seems chaotic both at 2D and 3D cases but at the same time streamlines are typically laminar.

V. INFLUENCE OF THE CHARACTER OF BREATHING TO SELF-CLEANING EFFECTS IN THE LUNG AIR WAYS

Lung modeling is very intriguing subject to investigate. It may be useful for investigation of respiratory ventilation systems, predicting of lung diseases, in developing an anesthesia simulator or other air drag delivery systems. For that purpose we need correct model to adequately predict temporal properties and space properties in the respiratory systems. The goal of this work is to develop model that is easy enough for use everywhere we want and can correctly predict spatial and time distributed properties. For that purpose we considered several levels of the scales for full physical modeling. Numerical simulation based on three dimensional unsteady Navier –Stokes equations is used for it. The possibility of wall's deformation is used, too.

This work is focusing in the lung modeling with particle deposition. Main attention is paid to the behavior of particles in a periodical (breathing) flow in the extensive system of channels.

The common pumps in human physiology have a several distinguish features from their technical counterparts. One of the features is the presenting multiple scales in the problem. The over features is a needs to pump fluid there and back periodically in deformable domain.

There are several commonly using ways to modeling aforementioned complex problem: full physical modeling, lumped modeling, distributed parameter modeling. All of them have their advantages and disadvantages. From one point of view, to be reasonable for use it must be easy enough. From another point of view it must capture necessary physical properties. For example, the lumped modeling and distributed parameter modeling are easy enough and can correctly predict changing in time. But it can not correctly predict spatial and time distribution simultaneously.

From the mechanical point of view lung appears as extensive network in which every branching doubles number of channels of smaller diameter dn. Total number of channels N=223 [1]. The first channel – trachea has d1 =18mm, the last – about 10 microns. Lengthening of channel varies lies between 6 and 1. These purposes generates basic problem of correct description of hydromechanics of a lung as a whole.

Method of our study is numerical simulation of three – dimensional flow of incompressible air in few first branches of lung. Flow describes by 3D Navier – Stokes equations. Model of walls allows to describe their deformation according some number of functions developed during the study. The task is formulated as solution of coupled problem of calculation of gas flow inside deformable boundaries.

VI. QUASI-PLANAR MODEL

Here the result of simulation of so-called "quasi-planar" model are described. In this model, first 4 levels of lung are modeled as 3D network but all axes of channels lay in one plane. Figure 6 depicts general and auxiliary coordinate systems used for correct description of wall's deformation. Mentioned problem is solved in the following way. First, local coordinate systems are placed at cross section of any channel modeled. Than, are used for specifying of wall's deformation of each channel.

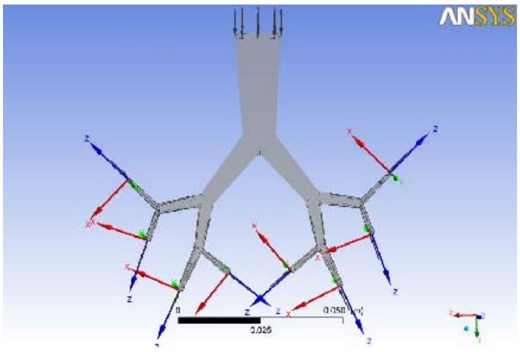
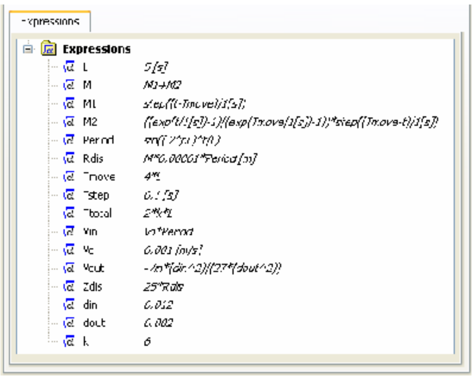


Fig.6 Simple boundary domain for lung modeling



```
Expressions
  L        5[s]
  M        M1+M2
  M1       step((t-Tmove)/1[s])
  M2       ((exp(t/1[s])-1)/(exp(Tmove/1[s])-1))*step((Tmove-t)/1[s])
  Period   sin((2*pi)*t/L)
  Rdis     M*0.00001*Period[m]
  Tmove    4*L
  Tstep    0.1[s]
  Ttotal   2*k*L
  Vin      Vc*Period
  Vc       0.001[m/s]
  Vout     -Vin*(din^2)/(27*(dout^2))
  Zdis     25*Rdis
  din      0.012
  dout     0.002
  k        6
```

Fig.7 Expressions for description of the domain deformation

Figure 7 depicts screen of ANSYS-CFX with user's expressions required for description of deformation. They are used at every time step. As example, the law of outlets displacement Zdis is placed on figure 8. Figure 9 depicts air velocity at the inlet of whole system (fig.6). Expression for outlet velocity is placed on screen's image (fig. 7). During breath and outward breath these laws used in reverse order. Laws of velocity and deformation are correlated.

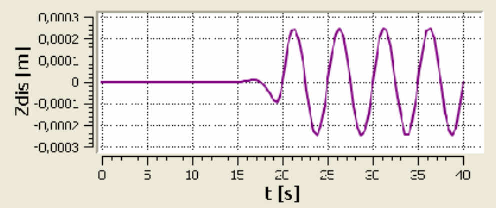


Fig.8 Law of domain deformation

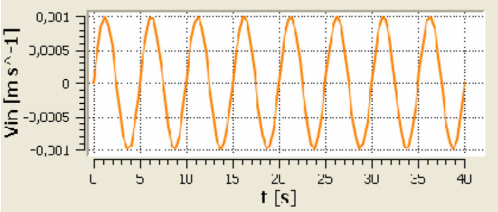


Fig.9 Inlet velocity

At the beginning of monitoring, the symmetric flow of air originates inside network (fig. 10a). Than, at some moment three-dimensional vortex originates in some branching. It accelerates flow in this channel. (fig. 10b). Finally, all flow rate concentrates in this channel with suction from adjacent channels, velocity in which trends to zero. System does not work as a pump more (fig. 11). It have to be noted that simulation without of wall's deformation gives the same result. It seems reasonably to assume that enlarging of number of channels may help to avoid such blocking.

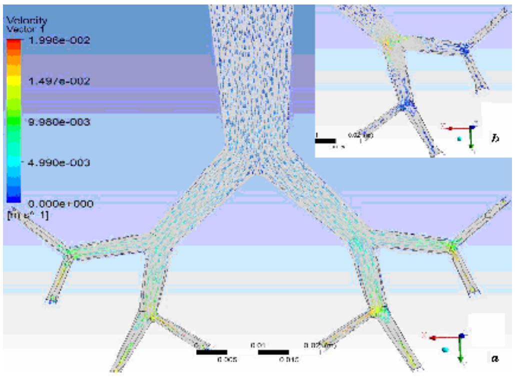


Fig.10 Vortex originates in some branching

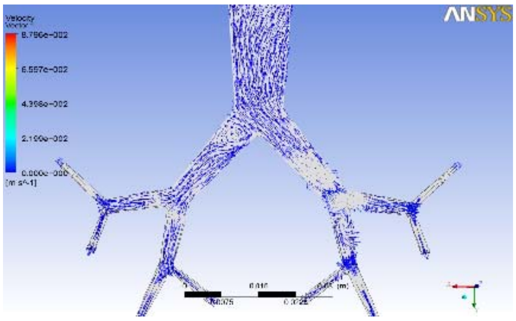


Fig. 11 System does not work us a pump more

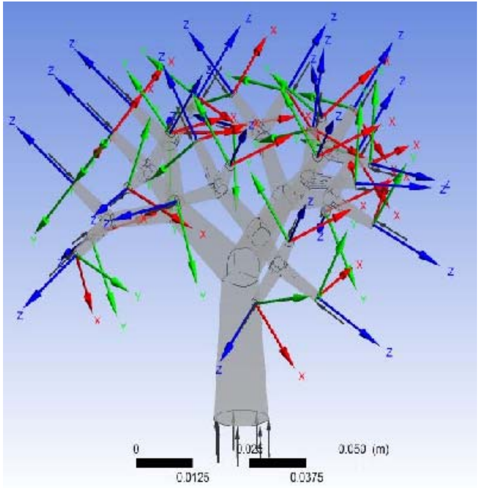


Fig. 12 Three-dimensional model

VII. Three – Dimensional Model

In accordance with above conclusion more complex model with 27 branches is studied. Axes of channels do not lay more in one plane but placed rather arbitrary in space (fig. 12). As first step, simulation without wall's deformation is realized. Distribution of velocity is shown on fig. 13. Inlet and outlet velocities are taken at the same manner as for quasy-planar case.

During the time of monitoring, particular attention is paid to distribution of average axial velocity V along arbitrarily taken sequence of cannels connected inlet and outlet of system. It appears that V increase in approaching of any brunch, decrease inside it and then increase again (fig. 13). Such qualitative feature realizes at any moment of time both on breath and outward breath.

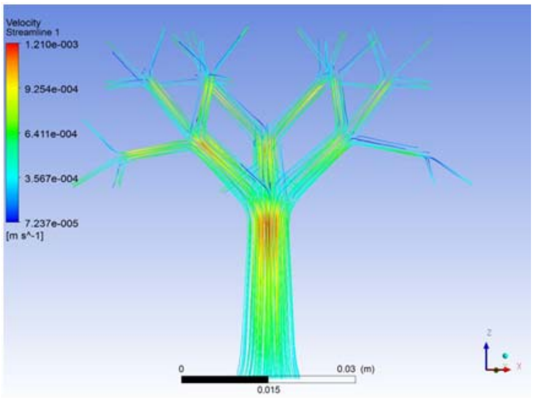


Fig.13 Velocity distribution

In the contrary with classical subsonic flow inside convergent – divergent channel, flow inside network of channels has an ability both accelerate and decelerate gas flow with changing in time of total area of inlet and outlet of channels involved into control. In other words, such geometry is “alive” because it shows possibility to self – regulation.

Another problem of essential importance researched during simulation is coordination of breathing and deformation of walls of system of channels. In order to solve it, the following experiment was done. After stabilization in time of flow pattern inside system under the study, the deformation of walls is switched on. Four laws of switching of deformation are studied: step-wise, linear, sinusoidal and exponential. It appears that only the last method ensures absence of mismatch of breathing and deformation. On fig. 14 the difference between maximal and minimal value of pressure inside whole system is depicted. From the beginning till 20 sec. flow is realized due to regulation of inlet and outlet velocities. After 20 sec. wall’s deformation is switched on exponentially. It is seen that it does not stopped breathing. This result seems useful for regulation of artificial lung and similar devices for lung ventilation.

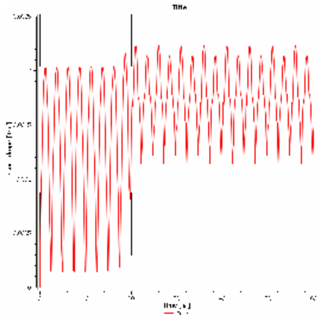


Fig.14 Pressure drop in the domain

In next step particle deposition was simulated using the lagrangian approach. Particles enter the domain with zero velocity slip. Diameters of the particles are 0.1 micrometer. Density of the particles is varied between air and water density. The calculations identified areas in which particles of different masses are beginning to gather in the caustic (fig. 15), which can then lead to the blocking of channels of air ways. In technical systems this can lead to erosion in these areas of channels.

On the breathe in and breathe out, the particles move at different speeds and acceleration over the section of the channel. You can also see that the particles with greater mass, in contrast to the light particles pass through the entire computational domain.

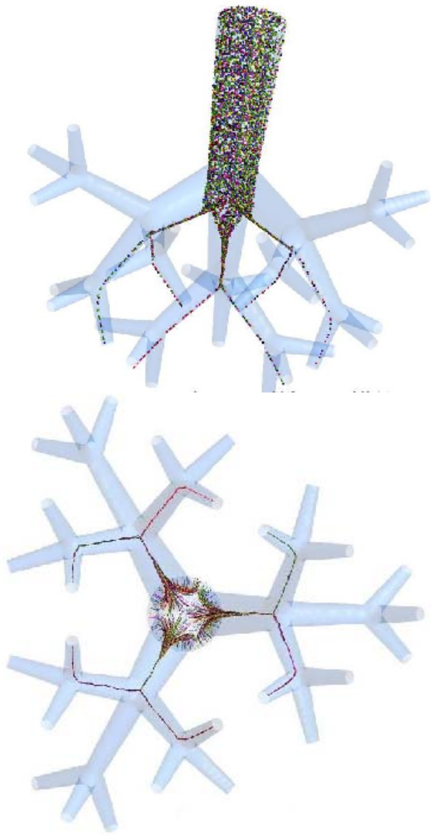


Fig. 15 Particles are gathered in the caustics

Depending on the ratio of drag force and gravity acting on the particle, and its location along the cross section, the particles of different masses behave differently. Particles during breathing out could remain in place or move.

The figure 16,17 are demonstrates the selfcleaning effects. We are looking to traveling one of the particles. In the phase of breathing out the particle velocity reduced by flow and between 13 -14 seconds became negative. This is giving to us an opportunity to use these effects for force the particles to moving out of the domain.

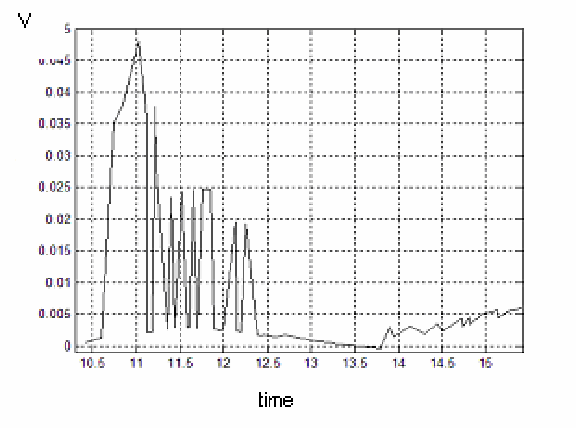


Fig. 16 Typical particle velocity along its tracking line

Figure 17 depicts the tracking line of considered particle.

The ball showing at the constant time intervals so we can see then and where particle reduce the velocity and so in that point we can force the particle to move out or change the traveling direction.

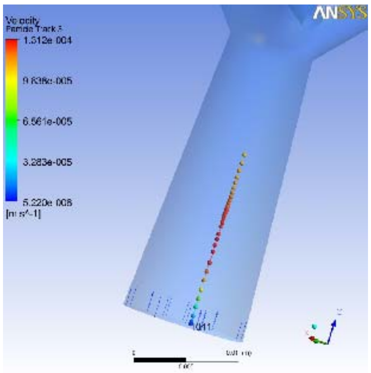


Fig. 17 Positions of the particle along its tracking line colored by velocity magnitude

The calculations found that the phase of breathing out can be used to clear the lungs. This calculation shows that this method would be more effective when used in combination with transverse vibrations of the channel walls.

REFERENCES

[1] Sturman R, Ottino J.M., Wiggins S. (2006). The mathematical foundations of mixing.

[2] Karniadakis G., Beskok A., Aluru N. (2005). Microflows and Nanoflows. Fundamentals and simulation.

[3] Num-Trung Nguyen (2008). Micromixers. Fundamentals, design and fabrication.

[4] Ottesen J. T., Olufsen M.S., Larsen J.K. (2004).Applied mathematical models in human physiology.

[5] Malestrom (2009)Lung Mechanics - An Inverse Modeling Approach