MAGNETOHYDRODYNAMICS Vol. 00 (1964), No. 00, pp. 1–5

LES modeling of heat and mass transfer in turbulent recirculated flows

E. Baake[1], *B. Nacke*[1], *A. Umbrashko*[2], *A. Jakovics*[2]

[1] *Institute for Electrothermal Processes, University of Hanover*
Wilhelm-Busch-Str. 4, D-30167 Hanover, GERMANY
baake@ewh.uni-hannover.de, nacke@ewh.uni-hannover.de
[2] *Laboratory for Mathematical Modeling of Environmental and Technological Processes, University of Latvia, Zellu str. 8, LV-1002 Riga, LATVIA*
andrey@modlab.lv, ajakov@latnet.lv

Experimental results show that heat and mass transfer processes in the turbulent melt flow of induction furnaces are significantly influenced by low-frequency large scale oscillations of the main flow eddies. Large Eddy Simulation (LES) of the turbulent melt flow in induction crucible furnace carries out with good conformity the transient three-dimensional oscillations of the dominating toroidal flow eddies. This 3D transient model offers new possibilities for simulation of heat and mass transfer processes in induction furnaces.

Introduction Fluid flow in industrial metallurgical installations has become a subject of numerical modeling many years ago. The variety of geometries, boundary conditions, fluid forcing factors and a whole range of flow Reynolds numbers provide a good challenge for numerical investigations. The absence of an universal and always reliable modeling approach together with a wide choice of available non-universal turbulence schemes turns it into a non-trivial problem, at least while up to now direct numerical simulation of the turbulent melt flow remains practically inapplicable. As a wide spread example can be mentioned the melting of alloys in induction furnaces. The flow pattern in these installations is formed by the influence of electromagnetic forces and usually takes form of several dominating vortices. Flow patterns obtained with two-dimensional solvers based on Reynolds Averaged Navier-Stokes (RANS) equations usually are in good agreement with estimated and measured time-averaged values [1-3]. Resulting spatial distribution of the temperature and alloys compound concentration depends strongly on the heat and mass exchange between these vortices. Numerical investigations show that two-dimensional turbulence models fail to describe correctly the heat and mass transfer processes between the main vortices. There was developed an engineering approach for this problem described in [1], but for more generic and therefore more flexible solution it is necessary to investigate advanced simulation methods. Today it is possible to run un-stationary three-dimensional numerical calculations of fluid dynamic problems using advanced turbulent models with higher grid resolution requirements and get reliable results in reasonable time. Concluding all these preconditions the calculations presented in this paper were based on Large Eddy Simulation (LES) method, which can be described as a compromise between the solving of RANS equations and Direct Numerical Simulation (DNS). The main idea is that main flow structure is resolved directly, while only small eddies, which size is comparable with grid size, are modeled additionally. Finer meshing is required and, consequently, more computational resources than for two parameter turbulence models, e.g. k-e turbulence model, but less than it is necessary for the DNS.

1. Low-frequency oscillations Most of the experimental and numerical investigations were carried out with a model induction crucible furnace (ICF), which has a radius of 158 mm and a height of 756 mm, where the inductor height is 570 mm (Fig. 1). Wood's metal, which has a melting point of 72^0C, and a dynamic viscosity of $4.2\cdot10^{-3}$ $kg/m\cdot s$, a density of $9,700$ kg/m^3 and a conductivity of $1\cdot10^6$ S/m was used as a model melt. Experimental velocity measurements carried out using a potential probe show presence of low-frequency flow oscillations in this model furnace (Fig. 2). Most intensive of them have a characteristic period about 8-12 seconds depending on inductor current I_{ind}. These oscillations can be described very simple as periodical up and down movements of the toroidal eddies of the mean melt flow, where the axis symmetry of the melt flow does not longer exist. The main oscillation frequency f increases in dependence on the time-averaged velocity: $f \sim v_{ch} \sim I_{ind}$.
As show experiments in model furnaces, when the characteristic flow velocity $v_{ch} \sim 0.20m/s$, then the maximum intensity of oscillations v'_{max} has the same order of magnitude ($\mid v_{ch} \mid \sim \mid v'_{max} \mid$). The maximum kinetic energy of the oscillations (in the zone near the crucible wall, between the main flow vortices) has value of $k^{exp}_{max} \sim 70cm^2/s^2$ (Fig. 3). These flow instabilities have a large characteristic time and space scale and therefore they can not be considered as turbulence. So it is doubtfully, that stationary calculations with help of any existing turbulence modeling approach could correctly simulate scalars transport processes in such flow formation. This statement was verified with several widespread turbulence models. All of them do not predict the high intensity of heat exchange in the discussed region. As expected, unstationary 2D calculations using k-e turbulence model also didn't lead to acceptable results. The absence of azimuthal velocity in 2D analysis limits the dynamics of the large eddies in incompressible fluid. Also 3D calculations using k-e turbulence model lead to not acceptable results because the low-frequency oscillations are dumped by the high turbulent viscosity. Transient 3D calculations appeared to be the only possible way to resolve the given problem.

2. Numerical modeling Transient calculations started with uninitialized flow field $\boldsymbol{v}(\boldsymbol{r}) \equiv 0$. After several seconds flow pattern comes to the symmetric state, which resembles results of 2D stationary modeling (Fig. 4b), and then begins to oscillate (Fig. 4c).
LES and two-equation RANS approach are using different turbulence modeling techniques, therefore, the calculated subgrid turbulent viscosity distribution has principally different character. The highest values of the LES subgrid viscosity are in the zone of eddies interaction in the near-wall region. Another unsimilarity is that turbulent viscosity in case of LES is one order of magnitude less than it is predicted by $k-\epsilon$ model. This difference increases if we improve the spatial resolution of the numerical grid. The smaller become cells, the less energy contain eddies, which are modeled with subgrid turbulent viscosity (Fig. 5a,c). Therefore, further mesh refining will lead to the situation when eddies of all considerable scales are resolved directly. In LES this tendency is obvious when examine the expression for subgrid turbulent viscosity μ_{sub} in Smagorinsky-Lilly scheme used in FLUENT [4, 5, 6]:

$$\mu_{sub} = \rho L_s^2 \mid S_{ij} \mid, \quad L_s = \min(kd, C_s \sqrt[3]{V_c}, \quad S_{ij} \equiv \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right),$$

where d is the distance from the closest wall, $k = 0.42$, $C_s = 0.1$ is Smagorinsky constant and V_c is volume of the computational cell. The averaged radius or cell size directly influences this expression. Kinetic energy of turbulence predicted with $k-\epsilon$ model doesn't depend on spatial resolution, and averaged velocity field doesn't change significantly with mesh refinement. Hence, velocity gradients, which are responsible for modeling of turbulent parameters, also remain the same. Certainly, if we use smaller cell sizes, then additional flow details may appear and influence the resulting turbulence field. But this dependence is not as obvious as in LES. For example, increasing number of mesh elements from 1/4 of million to 3.5 millions the subgrid viscosity calculated with LES becomes one order of magnitude less, but turbulent viscosity of $k - \epsilon$ model remains almost the same (Fig. 5b).
In $k-\epsilon$ simulation, due to high predicted turbulent viscosity, the viscous forces are comparable with inertial ones and, therefore, all possible large-scale pulsations are damped. Calculating with LES the subgrid viscosity is at least ten times smaller even on relatively coarse mesh. As result, oscillations, which arise from any disturbing factors, develop in time and reach significant amplitude.
There were chosen three control points for velocity along the crucible radius at half-height of the inductor, where the time-averaged velocity is approximately zero. Therefore, it was possible to analyze the oscillations and compare the time-depended behavior of the velocity components with the experimental results. Velocity oscillations obtained with transient numerical simulation using LES turbulence model are shown on the Fig. 6. Numerical simulation, using Smagorinsky-Lilly subgrid-scale model on tetrahedral mesh with $3.5 \cdot 10^6$ elements and with time step 10 ms, was taken as a base variant. Other simulations differ from the reference case just in one parameter: mesh resolution, time-step length or subgrid viscosity model. The flow development was calculated for 130 seconds on mesh with $0.4 \cdot 10^6$ elements, in other cases the simulated time-period was about 60 seconds.
Results of numerical modeling show, that the amplitudes of oscillations are in good agreement with those from measurements and their magnitude doesn't depend on number of elements. Being in good agreement with the experiment, the maximum of the oscillation intensity is located near the wall and decreases in the center of the furnace (Fig. 2, 3, 6). Following table lists the parameters of different numerical investigations and resulting characteristic values of low-frequency oscillations. The oscillations intensity in the given point is estimated by the time average of axial velocity square.

Mesh elements, $\times 10^6$	Subgrid turbulence scheme	Calculated time, s [time step, ms]	Oscillations intensity, cm^2/s^2	Low-frequency oscillation period, s
0.4	S-L	130 [10]	88	14
3.5	S-L	60 [10]	110	12
3.5	RNG	60 [10]	60	10.5
3.5	S-L	60 [5]	95	8.5
Experiment	-	58	122	9

Apparently we deal with a nonlinear system of Navier-Stokes equations, which has a solution tending to the limiting cycle. In other words, it is possible to estimate, with a good probability, the flow direction in a given point after relatively long time period. The numerical investigations show, that characteristic period of this main cycle doesn't change significantly when we improve the spatial or temporal resolution. Its length is determined by electromagnetic forces intensity

and domain geometry. Therefore, the initial conditions have no strong influence.
The discrete particle tracing approach has been carried out to investigate convective scalar transport mechanism in considered flow. In the very beginning virtual particles are placed on the top of computational domain. These particles are assumed to have the same density as fluid and this leads to the expectation that their path will coincide with the streamlines of the flow. When the flow in closed domain without inlets and outlets is stationary, the streamlines also are closed and particle trajectories should be looped. Then it is improbable that particle will penetrate into the neighboring flow region if the turbulent transfer is neglected. Therefore, transport processes between the main flow eddies generally would have diffusive character in steady-state flow. In this case scalar exchange intensity will strongly depend on the semi-empiric turbulent parameters like turbulent viscosity and turbulent Prandtl ($\delta_t = c_p\mu_t/\lambda_t$) or Schmidt ($Sc_t = \mu_t/\rho D_t$) number. Latter parameters magnitude often depends both on the type of fluid and on the type of the flow and has to be determined experimentally. The flexibility of using such approach for various industrial needs is rather low.
The subgrid viscosity model seems to have more universal character. Transient simulations, with a small time step and appropriate meshing, allow resolving the wide range of flow formations involved in scalar transport and also the anisotropy of the large scale turbulence is taken into account. Particle trajectories, traced during such unstationary calculations, show, that estimated convective mass exchange between the main flow eddies is quite intensive (Fig. 7a,b).
In our numerical simulations, four particles were launched simultaneously at $z = z_{max}$, $r = r_{max}/2$, $\Delta\varphi = \pi/2$ and traced for 20 seconds (about 5-6 eddy turnover times) in transient flow regime. The choice of starting position was caused by conformity with industrial alloying process, when additional components are added on the melt surface. From practical point of view, it is very usefull to be able to estimate the time, which is required for alloy's homogenization. The typical tracing result states, that particles usually don't stay in one eddy longer than two or three turnover times: at the end of simulation only one of four remained in the upper vortex. Therefore we can make the conclusion, that convective transport mechanism plays significant role in the heat and mass exchange between the main flow eddies. The same tracing procedure was used with the averaged velocity field from transient LES calculations. As expected, all particles rotated in the initial eddy with relatively small azimuthal drift (Fig. 7c). Probably, if we would take longer time for averaging, the trajectories would tend to those in steady-state flow pattern.

3. Conclusions Large Eddy Simulation turbulence model was applied for three-dimensional transient calculations and it proved to be a very promising tool for numerical simulation of complex turbulent flows. Produced results for velocity field and flow oscillations are in good agreement with experimental data. An extensive numerical analysis was performed to study how spatial and temporal resolution together with subgrid turbulence model choice influence solution stability and reliability. Particle tracing in transient flow showed the high efficiency of convective transfer due to simulated low-frequency oscillations. These results state, that, due to constant increasing of computational power, Large Eddy Simulation could become an universal tool for practical engineering applications.
Further numerical research in this area may include calculations with deformed surface of the melt in order to study its influence on the flow regime. Also the numerical simulation and optimization of the melting processes in industrial installations could be a logical continuation of the presented work.

REFERENCES

1. B. Nacke, E. Baake, A. Jackovics and A. Umbrashko. Heat and mass transfer in turbulent flows with several recirculated flow eddies. In *Fourth International PAMIR Conference*, (Presquile de Giens, France, 2000), p. 71.
2. R.M. Jones, A.D. Harvey and S. Acharya. Two-equation turbulence modeling for impeller stirred tanks.. *Transactions of the ASME*, vol. 123(2001), pp.640-648.
3. V. Bojarevics, K. Pericleous and M. Cross. Modeling turbulent flow dynamics in AC magnetic field. In *The 3rd International Symposium on Electromagnetic Processing of Materials*, (Nagoya, Japan, 2000), pp.85-90.
4. Fluent 5 User's guide. Fluent Inc. 1999.
5. J.S. Smagorinsky. General circulation model of the atmosphere. *Monthly Weather Review*, vol. 164(1963), pp.91-99.
6. D.K. Lilly. On the application of the eddy viscosity concept in the inertial subrange of turbulence. NCAR Manuscript 123(1966).
7. E. Baake. Grenzleistungs- und Aufkohlungsverhalten von Induktions-Tiegelöfen. VDI-Verlag, Düsseldorf, 1994.

Received March 14, 2003

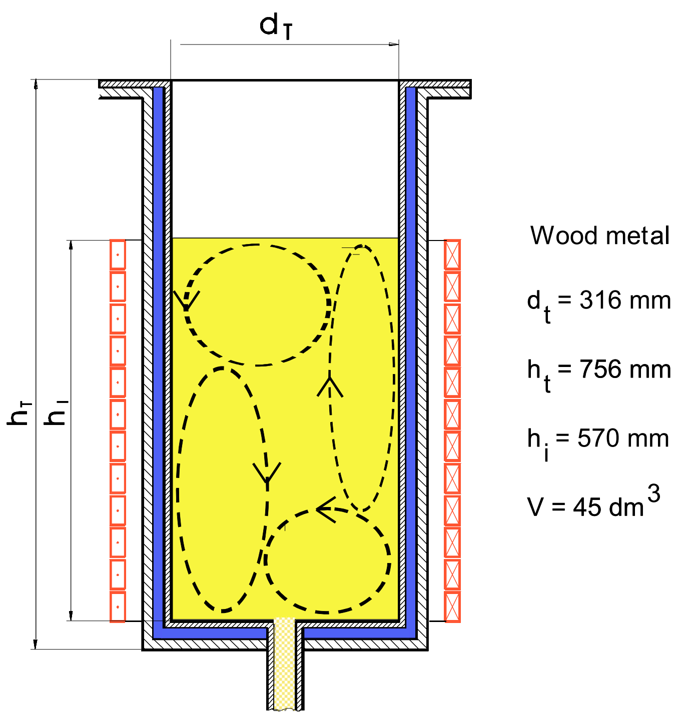

Figure 1: Experimental induction furnace used for 2D and 3D modeling. Characteristic flow directions and eddies' shapes are shown for the unstationary flow regime

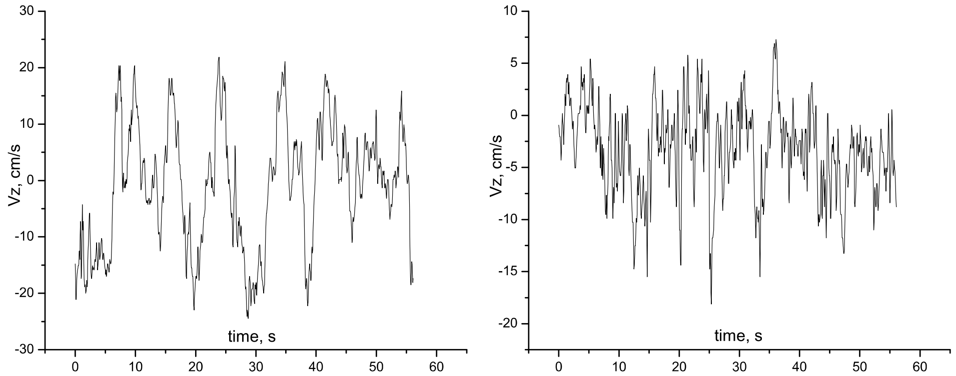

Figure 2: Measured axial velocity oscillations at the half-height of the inductor r=0.14m (left) and r=0.06m (right)

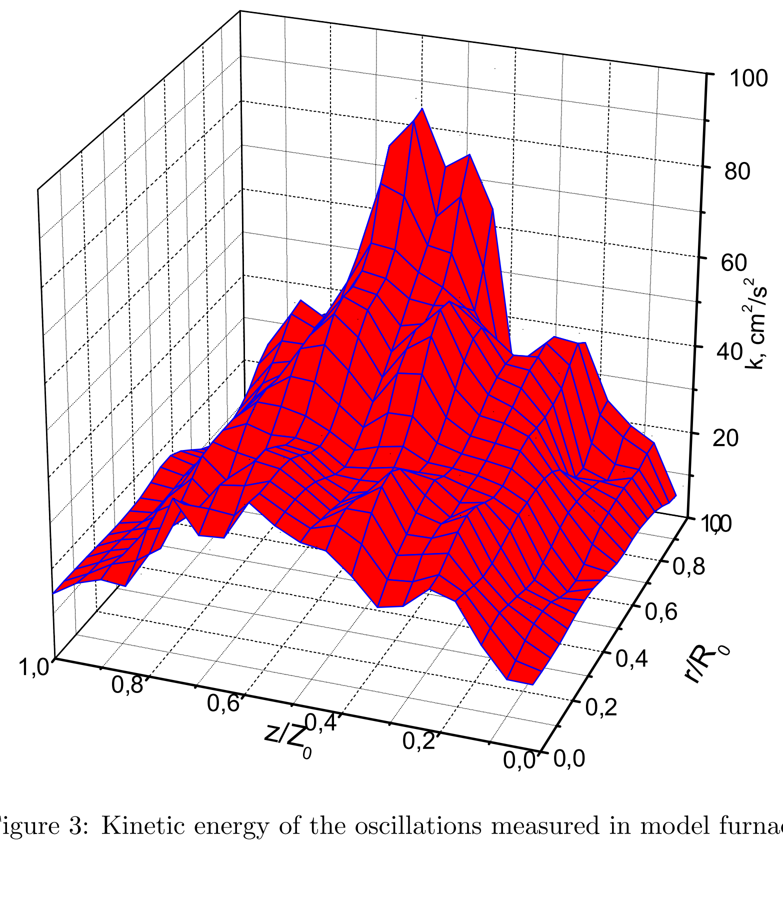

Figure 3: Kinetic energy of the oscillations measured in model furnace

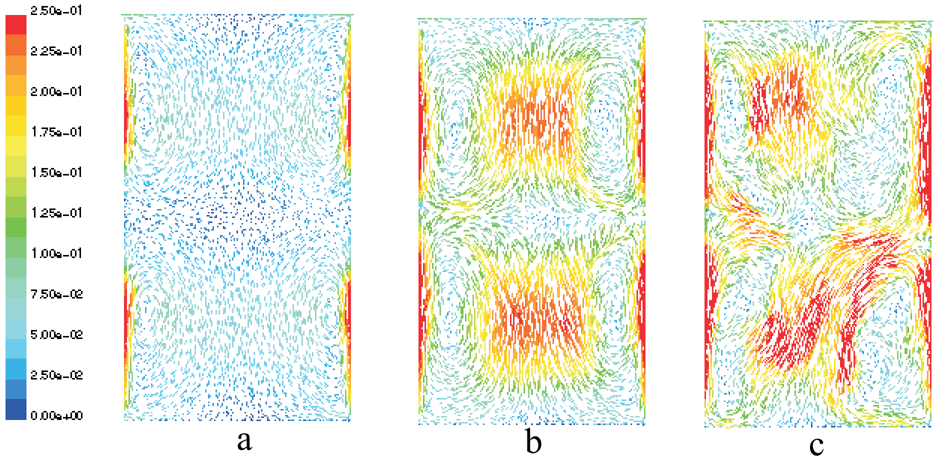

Figure 4: Flow pattern $[m/s]$ after 2, 6 and 10 seconds of calculations

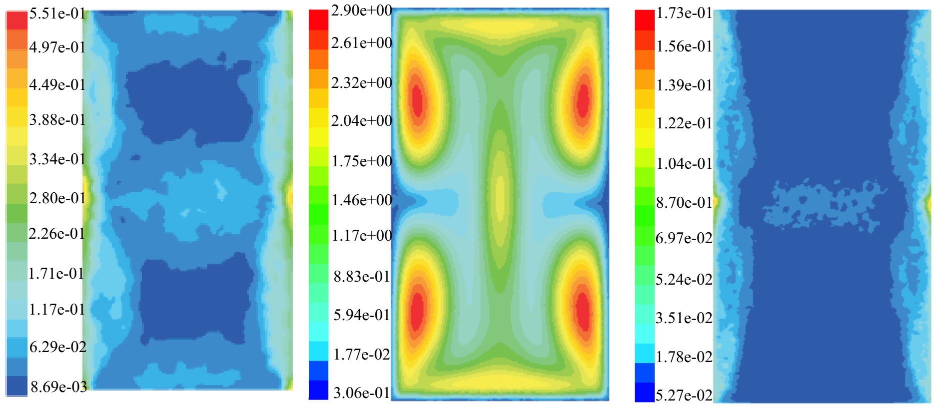

Figure 5: Turbulent viscosity distribution $[kg/m \cdot s]$ in case of $k - \epsilon$ (middle) and LES 3D modeling on mesh with $0.25 \cdot 10^6$ (left) and $3.5 \cdot 10^6$ (right) elements

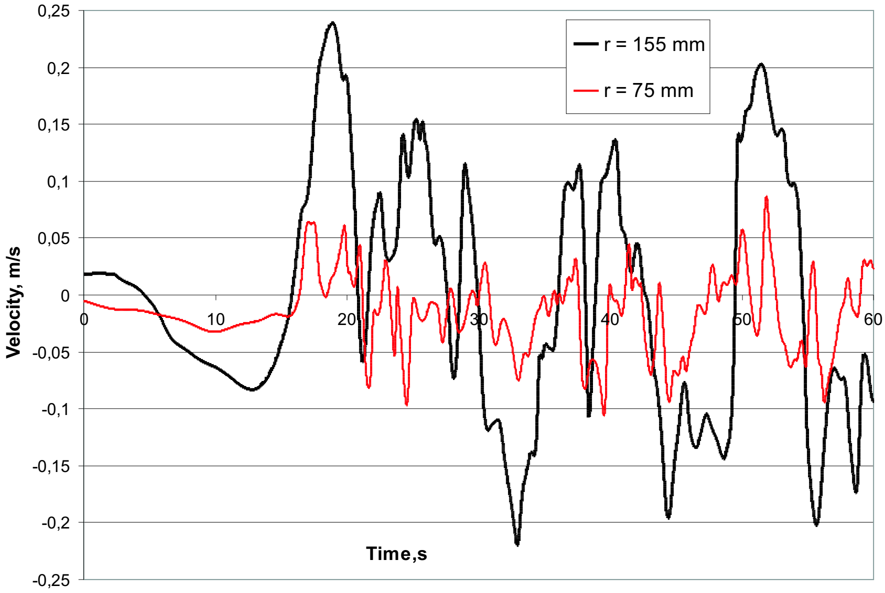

Figure 6: Axial velocity oscillations calculated with LES turbulence model on $3.5 \cdot 10^6$ elements mesh

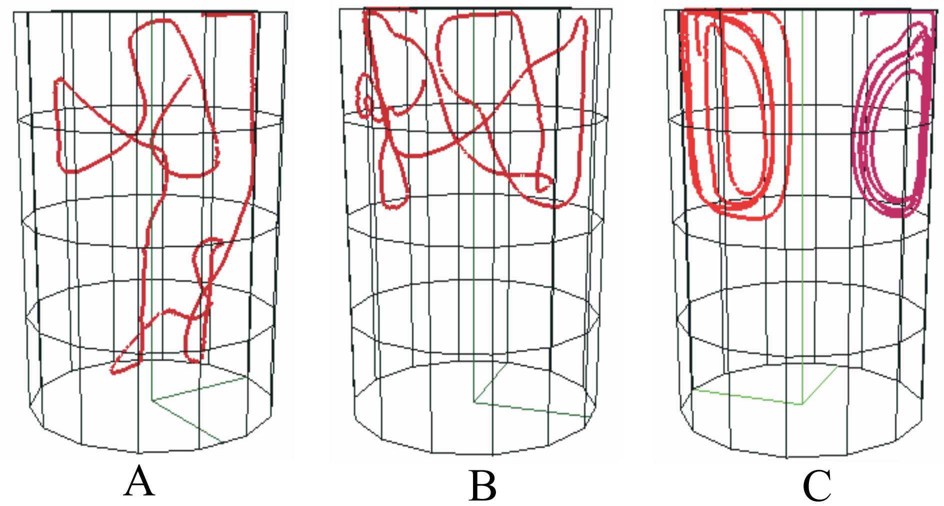

Figure 7: Results of particle tracing A, B in transient simulation and C in long time period averaged flow