VOL. 10, NO. 16, SEPTEMBER 2015 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

FOUR-DIMENSIONAL SIMULATION OF PARTICLE TRANSPORTATION IN SIMPLIFIED PASSAGEWAYS SYSTEM

Zamani Ngali[1], Nazri Huzaimi[2] Eng Pei Ying[3] and Kahar Osman[4]
[1,3]Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn MalaysiaJohor, Malaysia
[2]Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, Durian Tunggal, Melaka, Malaysia
[4]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Skudai, Johor, Malaysia
E-Mail: zamani@uthm.edu.my

ABSTRACT

Transient simulation of three-dimensional distribution of particles in fluid flow is one of the hardest challenge in Computational Fluid Dynamics field. This three-dimensional spatial solver plus temporal transient dimension that makes the whole four-dimensional analysis is more complex when it is formulated on non-geometric boundaries. This solver development work combines all these elements with aim of achieving a complete solver for constrained fluid-particle flows such as in pipelines, fluid passages, buildings or even more complex biomechanics geometries. The fluid solver is based on Splitting method while the particles are modeled with Lagrangian Particle Equation of Motion. The finite difference discretization is solved with semi-implicit technique. Simplified respiratory airway is used as case study to represent complex bounded fluid-particle flow. Temporal dimension plus spatial three-dimensional air and aerosol particles distributions are presented in 5 time frames. The simulation shows how the air flow evolved and particles deposited through the passage. Apart from fluid and particle flow pattern analyses, this work has also significant effect on mechanical analysis for instance particle-boundary collision momentum and collision density, hazardous consequences such as blockage, corrosion, abrasion and reduction in fluid passage efficiency.

Keywords: particle transportation, four-dimensional simulation, passageways system

INTRODUCTION

When dealing with particles in fluid passageways, engineers have always found that the transported particles through the passageways lead to serious problems to the piping system. This includes blockage by the particles deposition, corrosion and abrasion by the particles hammering on the passageway surface. These problems will harshly reduce the passageway efficiency due to loss of passageway integrity. The loss in transport efficiency and maintenance lead to greater loss to the overall company revenue. Thus, more sustainable approach is required to reduce the loss of revenue on design slip-up and maintenance.

An in-house four-dimensional solver for fluid and particle distributions developed earlier (Ngali et al., 2014) is an excellent alternative for this purpose. This sort of integrated solver is capable of simulating send deposition potential during the piping system design stage so that prevention and maintenance are optimized. Four-dimensional integrated fluid-particle solver is also suitable as passageway system evaluation tool to assist Oil and Gas practitioners to evaluate critical parameters such as piping size and particles minimum transport condition (MTC) during the system design stage.

There are numbers of schemes available when simulation of fluid-particles interrelations is of interests. One of the most popular and suit the requirement of Splitting Methods by (Karniadakis, 1991) which is used as the fluid solver in this work is Eulerian-Lagrangian scheme. Among others, (Patankar and Joseph, 2001) are the pioneers to commence the application of Eulerian-Lagrangian technique in dealing with fluid-particle interactions.

Nomenclature

F_E	External Force
F_B	Buoyant Force
F_D	Drag Force
C_D	Drag coefficient
A_p	Projected area of particle
$N_{Re,p}$	Particle Reynolds Number
Dp	Diameter of the particle
ρ_f	Density of fluid
μ	Viscosity of fluid
g	Gravity
ρ_p	Density of particle

Particle distribution solver is widely developed based on the Particle Equation of Motion formulation. The same concept is adopted in this work. The validation of fluid-particle solver coupling is focusing on the particle conformity characteristic. This attribute is vital in various phenomena such as sedimentation, deposition, ventilation and waste management.

Despite the fact that there are only limited academic resources discussing the characteristics and behaviours, there are few works have been done on particle conformity to fluid flow in the past few years. (Kosinski *et al.* 2009) among others, made a qualitative comparison of a single solid sphere particle path in a lid-driven cavity between his numerical solutions with experimental results by (Tsorng *et al.* 2006).

Another important parameter need to be considered for the simulation of fluid-particle distribution in passageway system is the gravitational force. An important introduction of gravitational effect feature in Splitting Solver was made by (Ngali *et al.* 2014) where the aim was to improve the solver feasibility upon environmental air-particle related case studies. Even though the air and particle distribution analyses were done mainly for environmental studies such as dust ventilation, aerosol control or granular fertilizer blower distributions, the same solver is expected to be able to simulate this work's interest on passageway system flow distribution.

Wrapping up, this solver development work combines the importance of gravitational effect, boundary interactions and transient distributions both on fluid and particles distributions through passageways.

RESEARCH METHODOLOGY

Particle Equation of Motion is used for the particle trajectory. If particle of mass m moving through a fluid under the action of an external force FE while the velocity of the particle relative to the fluid is u, the buoyant force on the particle is FB and the drag is FD, then we have,

$$m(du/dt) = FE - FB - FD \qquad (1)$$

Where,

$$FD = CDu2\square Ap/2 \qquad (2)$$

where CD is the drag coefficient, Ap is the projected area of the particle in the plane perpendicular to the flow direction. Drag coefficient is a function of Reynolds number, denoted as,

$$NRe,p = (uDp\square) / \mu \qquad (3)$$

Where u is the velocity of approaching stream, Dp is the diameter of the particle, □ is the density of fluid and μ is the viscosity of fluid. The drag coefficient is obtained by solving,

$$CD = 24/(NRe,p) \qquad (4)$$

Validation of the developed three-dimensional fluid-particle integrated solver in complex geometry was established with a simplified human airway case study similar to simulations made by (Zhang *et al.* 2002) as shown in Figure-1. The current work however was carried out for more complex unsteady analysis rather than a steady solution. The particle flow simulations were made in parallel with the fluid flow time iterations unlike the setup by (Zhang *et al.* 2002) where the particles were simulated separately with an invariable steady velocity distribution.

While the previous work was done using CFX4.4 commercial software and its combinations with user-support programs, the current results were all produced using this single fully integrated algorithm that consist of pre-processing, fluid-particle solvers and post-processing codes.

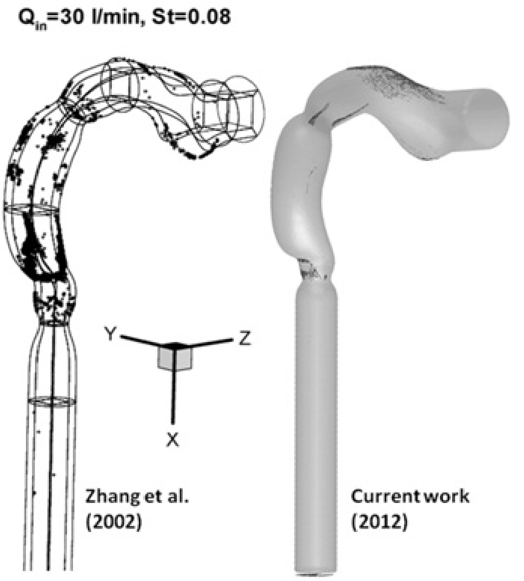

Figure-1. Particle deposition comparison between Zhang *et al*., (2002) at left and current work at right.

RESULTS AND DISCUSSIONS

Figure-2 illustrates the snapshots of transient velocity contours for six different time intervals. In a good agreement with velocity contour reported by (Zhang *et al*., 2002) for the convergence to a steady state condition, this current work found that there was an appealing transient flow condition for this setup as 30 l/min flow rate was dashed to reach steady state flow condition within this level of geometrical complexity.

Referring to the figure, the first pair at time 0.0001 was the initial condition where velocity was set to zero at all locations and 10,000 homogeneous particles with Stokes number equivalent to 0.08 were located randomly at the mouth cavity opening, similar to setup by (Zhang *et al*. 2002). The velocity inlet and outlet were placed equivalent to 30 l/min flow rate. The airway wall was arranged to be half visible to allow more sensible visualization of the centre slice velocity contour. The overall array employed by the simulation was 123x206x37 which was equivalent to 937,506 numbers of nodes.

The effective calculated nodes however were only 83,191 where the remaining numbers were considered as null nodes with no calculations required. These computational nodes were less than a quarter of the nodes used by (Zhang *et al.* 2002) where 480,000 were recorded. The right figure wall was also set as 70 percent translucent to allow full visualization of particles moving through the passageway. The flow time increment was set to be 0.0001 while the particle time increment was at 0.00001 which indicated exactly 10 inner particle iterations in every flow iteration.

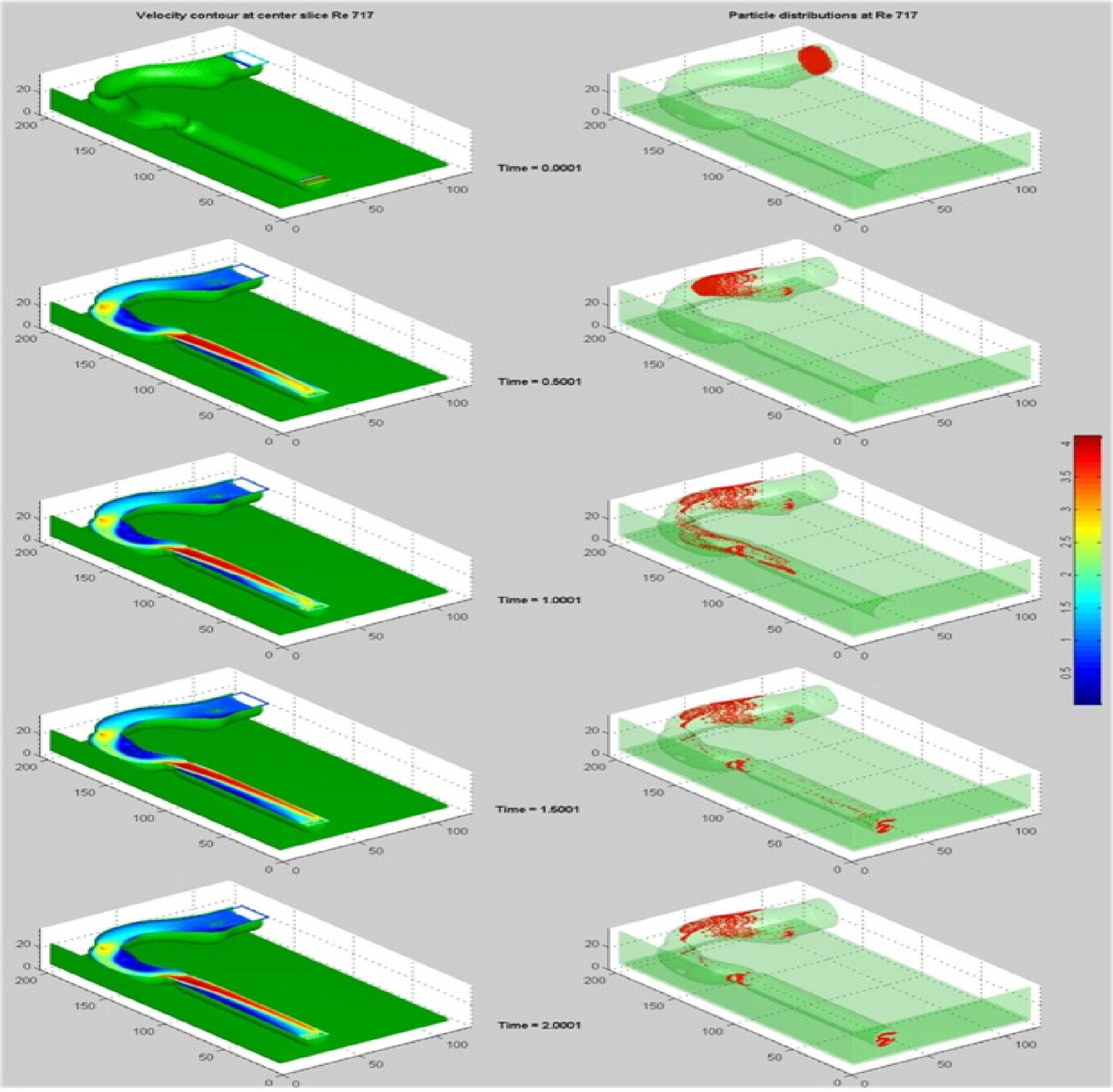

Figure-2. Transient velocity contours and particle distributions for simplified upper human airway system.

At time 0.5001, the velocity contour noticeably indicated a transient flow behavior located right after the glottis throttling area. Vortex formation at the low flow zone was seen at its initial stage while flows at mouth cavity down to pharynx region were already in steady condition. As for the particle distribution, particles motivated at the centermost region were observed having the maximum velocity and reached nearly the end of mouth cavity at this particular time. More than a few particles were deposited at the tongue side of center of mouth cavity. The remaining particles were still passing through the passage with increasing velocity towards the first constriction of the airway system as the cross sectional area reduced toward the back side of the oral cavity.

In contrast with the result by (Zhang *et al.* 2002), wall surrounding the area entering the soft palate was observed to be the location most hit by passing particles. This can be seen starting from figure representing time of 1.0001 onward. At this particular time, velocity contour was also found to flow in a consistent fluctuation due to the jet effect at glottis area while flows before this maximum velocity area maintained its steadiness. Within the same period, none of the particles found to reach the trachea exit where the most advance particles were still at the middle of the trachea duct.

Figures representing the time frame of 1.5001 and 2.0001 verify the decaying pattern of the flow fluctuation due to the glottis throttle. Further observation for the duration beyond those illustrated in this figure also showed that the flow fluctuation pattern was only a transient response and diverging into steady state condition similar to the simulation by (Zhang *et al.* 2002). While the velocity contours showed insignificant differences on the last two time frames, these five sets of snapshots were selected purposely for the particle distribution visualization over time up to the period where no particles were still in motion.

Once done, particles were found deposited mostly at locations with low flow zones away from the main stream pathway. There was no significant deposition at locations where the flow struck the wall surface as predicted. Meanwhile for other locations, minimum depositions were found at pharynx and trachea regions. After all, only 26.48 percent from the total 10,000 particles were manage to reach the trachea end while 59.04 and 14.12 percent were found deposited at the oral cavity and glottis entrance respectively. Only the remaining 0.36 percent of the total number of particles was found deposited elsewhere.

Qualitative comparison between the velocity contour results at steady state by (Zhang *et al.* 2002) and the current results at time 2.5001 can be found in Figure-3. Even though the exact match of the contour values at all nodes were not possible due to un-identical model construction, solver degree of accuracy and even the grid intensity deviations, the flow patterns as shown in Figure-3 were comparably of the same kind. Locations where supposedly at maximum, minimum and main streamline velocities were all matched with superbly indistinguishable velocity ratio values.

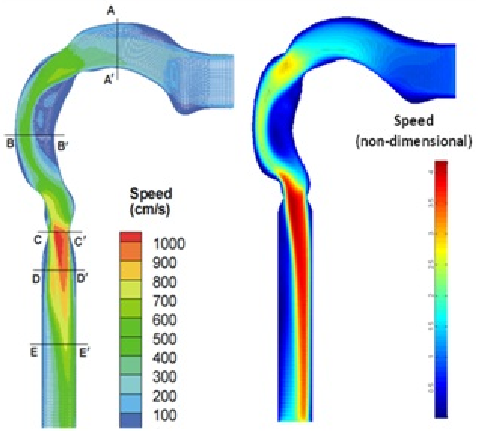

Figure-3. Velocity contour comparison between Zhang *et al.* 2002 at left and current work at right for flow rate equivalent to 30 l/min.

CONCLUSIONS

The main objective of this work which is to obtain a transient simulation tool for three-dimensional distribution of particles in fluid flow formulated on non-geometric boundaries has been successfully accomplished. The application of in-house four-dimensional fluid-particle integrated solver was proven to be useful especially in comparing the particle distributions at any given instances. The importance on such simulations in design, evaluation and maintenance stages of fluid-particles in passageways system preservation become more significant when it deals with turbulent flows. From the analyses carried out in this work, we can conclude that the developed in-house integrated solver is capable of simulating fluid-particle distribution within passageways system. The simulation results are capable of forecasting the potential hazards the pipeline may have even before the system is constructed. By simple calculation and analysis on the overall particle collision momentum and collision density, hazardous consequences such as blockage, corrosion, abrasion and reduction in pipe efficiency can be predicted accordingly.

Large industries such as petroleum upstream companies and agricultural particles distributions could benefit from this simulation tool with further advancement on the computational capabilities. Large scale equipments with huge demanding flow criteria acquire suitable upgrading on the current solver especially on computational memory management, boundary layer formulation and crunching efficiency.

ACKNOWLEDGEMENTS

This research is supported by the Fundamental Research Grant Scheme (FRGS), Grant No: FRGS/2/2013 Vot 1420 by Ministry of Education, Malaysia.

REFERENCES

[1] Karniadakis G., Israeli M. and Orszag S. 1991. High-order splitting methods for the incompressible Navier-Stokes equations, Journal of Computational Physics, Vol. 97, pp. 414-443.

[2] Kosinski P., Kosinska A. and Hoffmann A.C. 2009. Simulation of solid particles behaviour in a driven cavity flow, Powder Technology journal, Vol. 191, pp. 327–339.

[3] Ngali M. Z., Zakaria N. H., Kahar O., Khalid A., Manshoor B. and Zaman I. 2014. Gravitational Effect Formulation on In-House Air-Particle Flow Solver, Applied Mechanics and Materials, Trans Tech Publications, Switzerland, Vol. 660 pp. 699-703.

[4] Patankar N.A., Joseph D.D. 2001, Modeling and numerical simulation of particulate flows by the Eulerian-Lagrangian approach, International Journal of Multiphase Flow, Vol. 27, No. 10, pp 1659-1684.

[5] Tsorng S. J., Capart H., Lai J. S., Young D. L. 2006. Three-dimensional tracking of the long time trajectories of suspended particles in a lid-driven cavity flow, Experiments in Fluids, Vol. 40, pp. 314–328.

[6] Zhang Z. and Kleinstreuer C. 2002. Transient airflow structures and particle transport in a sequentially branching airway model. Phys. Fluids. Vol. 14, pp. 862–880.