Smoothed particle hydrodynamics simulations of flow in air diffuser

Cite as: AIP Conference Proceedings **1768**, 020006 (2016); https://doi.org/10.1063/1.4963028
Published Online: 23 September 2016

Tomas Korinek, and Karel Frana

Smoothed Particle Hydrodynamics simulations of flow in air diffuser

Tomas Korinek[1,a)] and Karel Frana[1,b)]

[1]*Department of Power Engineering Equipment, Faculty of Mechanical Engineering, Technical University of Liberec, Studentska 2, 461 17*

[a)]Corresponding author: tomas.korinek@tul.cz
[b)]karel.frana@tul.cz

Abstract. This study is focus to the Smoothed Particle Hydrodynamics (SPH) method for simulations in indoor environment quality problem. There were done a few benchmark cases for better understanding of this method and for a validation of SPH program DualSPHysics. The benchmark test was flow past square cylinder, where different Reynolds number were tested. The main case was an observation of airflow through a simple air diffuser. Results from numerical simulations where SPH method were used were compared with results from numerical simulations where finite volume method (FVM) with Reynolds Averaged Navier-Stokes approach (RANS) was used. There were compared instantaneous velocities obtained from numerical simulations of both methods (FVM and SPH). Numerical simulations were done using classical processor (CPU) and graphic card unit (GPU).

INTRODUCTION

The SPH was first introduced by Monaghan [1] to deal with astro-dynamical problems. SPH is a mesh less method, which uses a Lagrangian approach for simulations in fluid dynamics. This method is well used for a simulation of free surface flow. The SPH method enables a huge parallelization and is suitable for a calculation on graphic cards (GPU). A great benefit of using GPU for a calculation is enormous number of cores in one GPU compared with classic processor (CPU). Nowadays it is possible to put 4 GPU to one personal computer under Windows and 8 under LINUX.

The most of studies [2-6] are focused on a simulation of the free surface flow, where the SPH method is well suited. The SPH needs smaller time step size in comparing with FVM [3]. Some studies which using SPH are focus on multiphase flows [7-8]. There have been proved, that multiphase simulation takes more than five times longer than a single phase flow. Capabilities of using the SPH for a simulation of a phase-change process like melting process were investigated a good agreement with experiment [9]. Simulations of reactive transport require a fine mesh and a small time step size in case of using FVM. Simulate this phenomenon with the SPH method was investigated on case of surface growth due to precipitation/dissolution and chemical reactions [10]. A change of particles resolution was also investigated [11] and it is done by splitting a large particle to smaller particles. This improves capabilities of the SPH for domains with variety shapes.

There was used DualSPHysics [12] program for numerical simulations. This program allows computing on CPU and also on GPU. Numerical simulations in this study was done on NVIDIA GeForce GT 740M graphic card (384 cores), where the speedup was 6.7 in comparison with calculating on CPU Intel Core i7 4770. The graphic card NVIDIA GeForce GTX 780TI with 2880 cores had speedup almost 50. The speedup depends on number of computed particles. The speedup rises with number of particles until the point of saturation, after that the speedup is constant. However the speed up was achieved on single precision. It has been found, that 4 millions of particles need about 1 GB of memory on GPU with DualSPHysics program. There are some operations which are done on CPU, if user choices the GPU solver. Initialization and saving output data are done on CPU.

The Meeting of Departments of Fluid Mechanics and Thermomechanics (35MDFMT)
AIP Conf. Proc. 1768, 020006-1–020006-7; doi: 10.1063/1.4963028
Published by AIP Publishing. 978-0-7354-1426-6/$30.00

PROBLEM FORMULATION

Mathematical model

The SPH method is based on a simulating of particle interaction. In SPH are simulated forces that act on each particle. A measure of influence of particle b to particle a is weighted by a weighting function W called kernel function. The first used kernel function for SPH was Gaussian function, but nowadays Wendland and cubic spline are most used [13]. The Cubic spline kernel function and its derivate is shown on Fig. 1 right. The criterion which particles will be taken in consideration during computing forces on particle a is a smoothing length h, this is shown on Fig. 1 left.

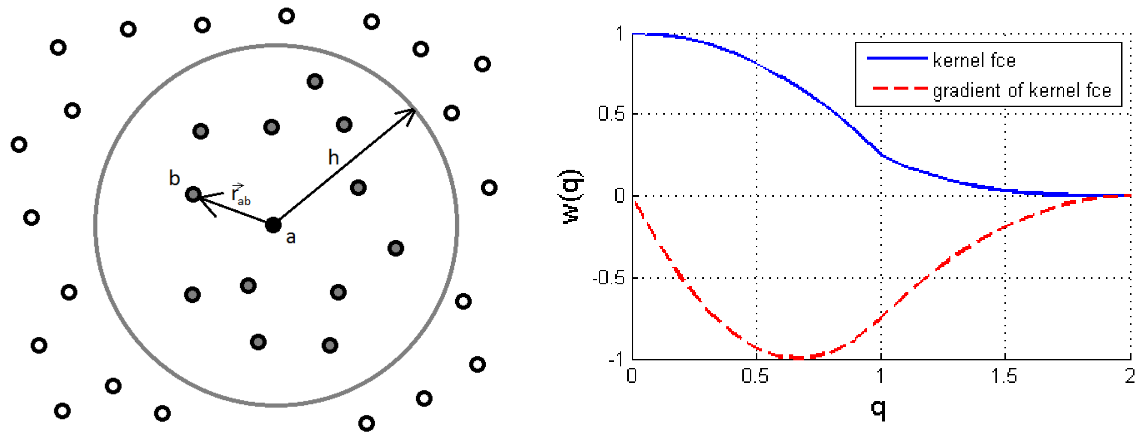

FIGURE 1.The influence of particles b to the computed particle a (left), Cubic spline kernel function.

Kernel functions are basis of the SPH and choice of the right kernel function is the main task. The function of cubic spline kernel (1) is shown below, where q is non-dimensional distance between particles q=r/h.

$$
\begin{aligned}
w(r,h) &= \alpha\left(1-\frac{3}{2}q^2+\frac{3}{4}q^3\right) \quad 0 \le q \le 1 \\
w(r,h) &= \alpha\frac{1}{4}(1-q)^3 \quad 1 \le q \le 2
\end{aligned} \tag{1}
$$

Where α is $10/7\pi h^2$ for 2D and $1/\pi h^3$ for 3D. A mathematical model contains continuity equation (2) and momentum equation with laminar viscosity and Sub-Particle Scale (SPS) turbulence (3).

$$
\frac{\partial \rho_a}{\partial t} = \sum_b m_b \vec{v}_{ab} \cdot \nabla W_{ab} \tag{2}
$$

$$
\frac{\partial \vec{v}_a}{\partial t} = -\sum_b m_b\left(\frac{P_b}{\rho_b^2}+\frac{P_a}{\rho_a^2}\right)\nabla W_{ab} + \sum_b m_b\left(\frac{4\nu_0 \vec{r}_{ab}\cdot\nabla W_{ab}}{(\rho_a+\rho_b)\left|\vec{r}_{ab}\right|^2}\right)\vec{v}_{ab} + \sum_b m_b\left(\frac{\vec{\tau}_b}{\rho_b^2}+\frac{\vec{\tau}_a}{\rho_a^2}\right)\cdot\nabla W_{ab} + \vec{g} \tag{3}
$$

Index a denotes computed particle and b denotes neighboring particle, ρ is density, m is mass of the particle, P is pressure, ν_0 is kinematic viscosity, τ is sub-particle stress tensor, g is gravity acceleration, v_{ab} is a relative velocity between particles a and b, r_{ab} is a vector between particles a and b. The pressure of the particle is calculated using the Tait equation of state (4).

$$
P = B\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right] \tag{4}
$$

Typical expression for $B=c_0\rho_0/\gamma$, for water is γ=7, c_0 is speed of sound and ρ_0 is reference density.

Benchmark test setup

The benchmark test was a flow past a square cylinder, where the benchmark domain is shown on Fig. 2. The three different Reynolds number were tested (Re_1=3750, Re_2=9375. Re_3=18750), where the Reynolds number was defined as $Re=(d \cdot v)/\upsilon$ and the length scale d was 0.25 m. The fluid was air with the density ρ=1.2 kg/m^3 and the kinematic viscosity $\upsilon=1.5 \cdot 10^{-5}$ m^2/s. The initial time step size for SPH was $1 \cdot 10^{-5}$ s and minimal was $1 \cdot 10^{-6}$ s, where Courant Friedrich Lewy number was set on CFL=0.2. Number of particles in domain was 98890. The inlet was on the left side and outlet on the right side.

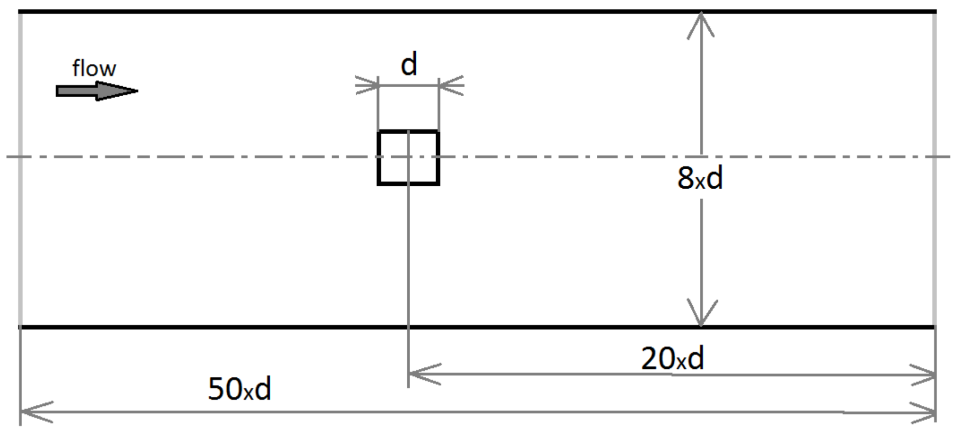

FIGURE 2.The benchmark domain.

Air diffuser setup

The simple air diffuser with 8 blades was investigated, where the geometry is shown on Fig. 3. The velocity of air before the diffuser was 0.05 m/s. The properties of air were the density ρ=1.2 kg/m^3 and the kinematic viscosity $\upsilon=1.5 \cdot 10^{-5}$ m^2/s. The diameter of particle was set on $dp=5 \cdot 10^{-4}$ m, this value was also set as a minimal length of element for FVM. The initial time step size for SPH was $1 \cdot 10^{-5}$ s and minimal was $1 \cdot 10^{-7}$ s, where Courant Fridrich Lewy number was set on CFL=0.1. Number of particles in domain was 635526.

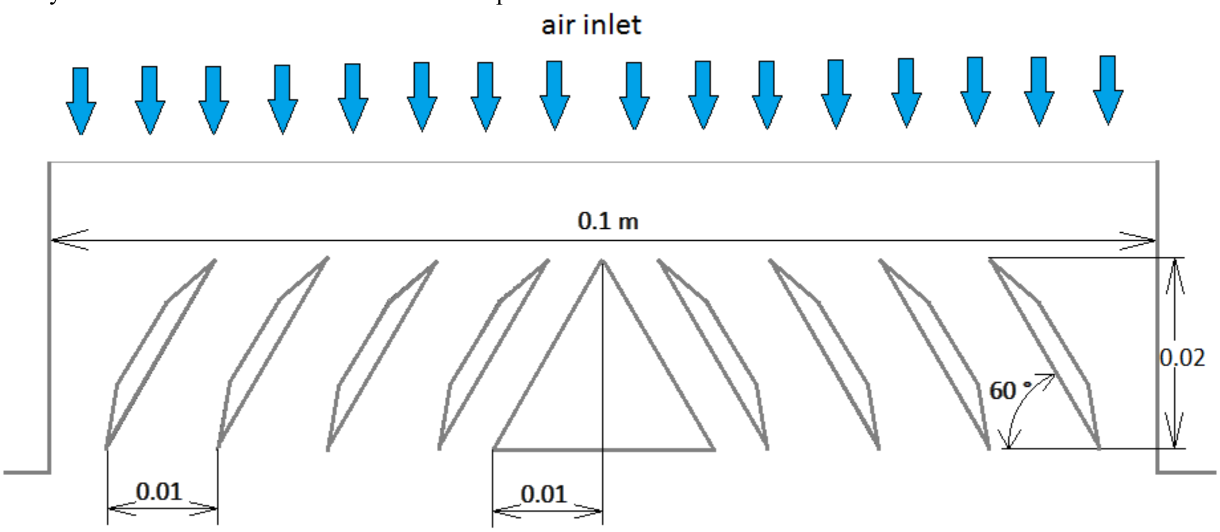

FIGURE 3.The geometry of the air diffuser.

The air diffuser was only numerically investigated. There was not done any experimental measurement on the diffuser. The main task was to analyze, whether the SPH is suitable for the indoor environment quality problem.

RESULTS

The first test was the validation on the benchmark test. There is a comparison of x component of velocity on Fig. 4 between SPH and FVM. A good agreement was achieved for the x velocity component in all cases. There was a higher velocity from SPH simulation behind the square cylinder in comparison with FVM for Re=9375. This difference was also for other Reynolds number, but it was much smaller than for Re=9375.

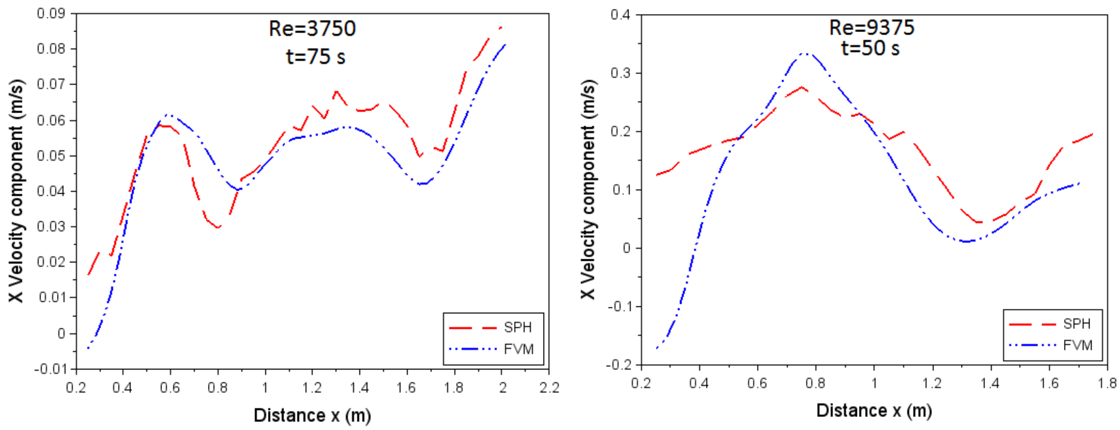

FIGURE 4.Comparison of x velocity component behind the square cylinder in axis of symmetry between SPH and FVM, for Re=3750 in time 75 s (left), for Re=9375 in time 50 s (right).

Velocity contours are shown on Fig. 5, where upper contours are for FVM and lower contours are for SPH. Contours from the SPH simulation are in a good agreement with contours from the FVM simulation.

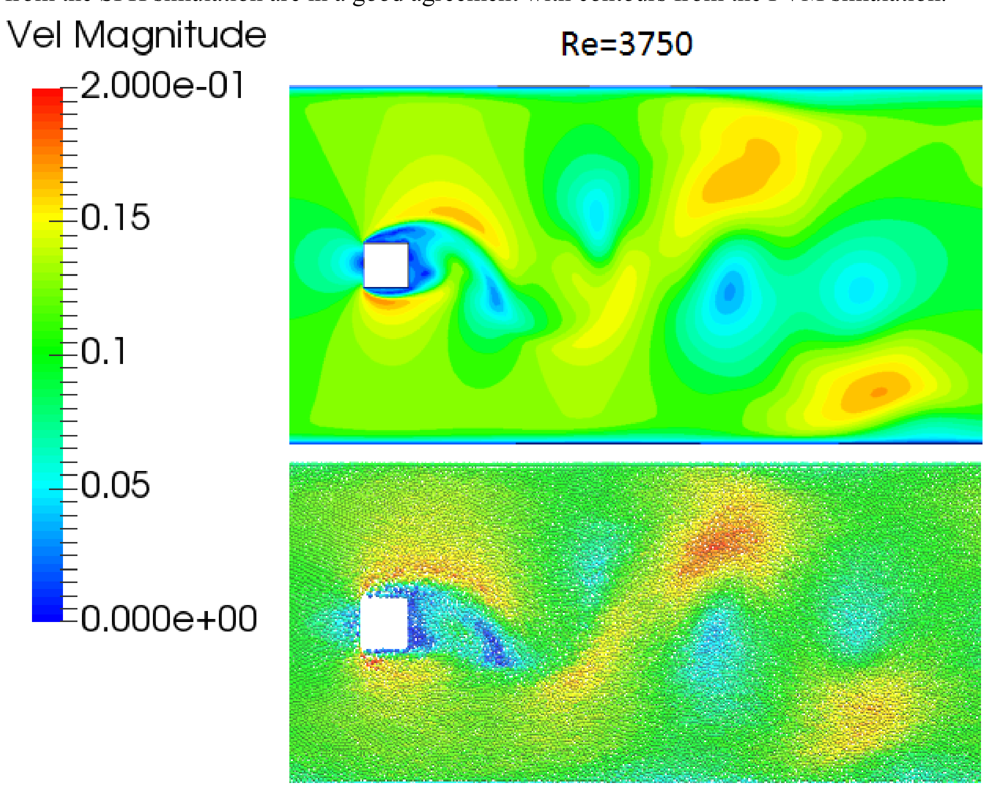

FIGURE 5. Contours of velocity for Re=3750 in time 75 s, FVM(up) and SPH (down).

There were tensile instabilities for higher Reynolds number. Instabilities were created by a separation of boundary layer near a solid wall and slowly disappeared in the flow. There is a creation of instability on Fig. 6. Number of instabilities raises with increasing of velocity in domain. This could be by inappropriate boundary condition for pressure on the wall. Also DualSPHysics does not compute with absolute pressure. This is because DualSPHysics 3.2 is primary for the free surface flow and the source code is optimized for this problem. This has been optimized in new release using the shifting algorithm [5].

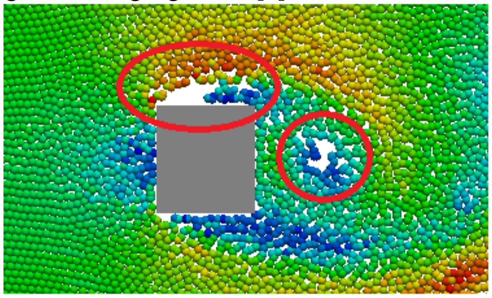

FIGURE 6. Tensile instabilities for Re=9375.

The second case was airflow through the air diffuser. Results from the SPH simulation were compared with results from the FVM simulation. There is a comparison of velocity contours between FVM and SPH on Fig. 7. Differences in contours are negligible There was not applied any filter or smoothing function for results from SPH and displayed data were colored by the value of velocity of particles.

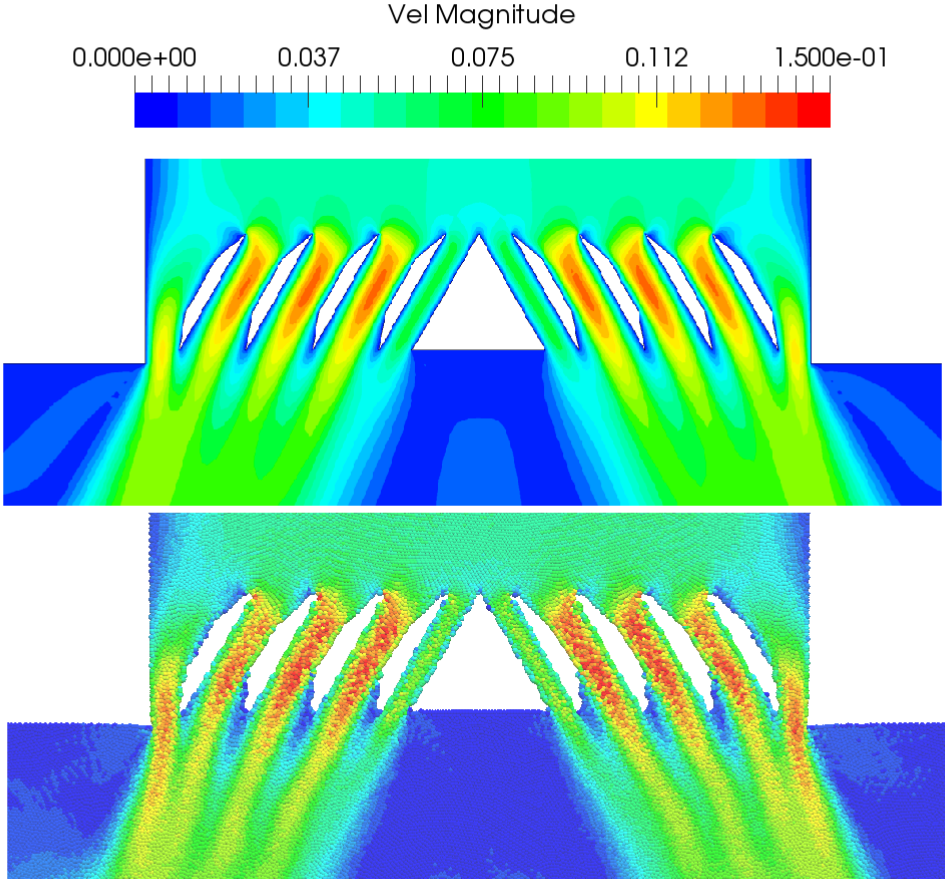

FIGURE 7. Contours of velocities inside the diffuser for FVM (up) and SPH (down).

There were compared velocity profiles between two blades of the air diffuser. The velocity profile from the FVM simulation is more smoothed than profile from the SPH simulation. This could be caused by number of particles in the SPH simulation. More particles between blades mean a smoother velocity profile. DualSPHysics does not contain any refinement function, which could help with decreasing of number of particles, yet.

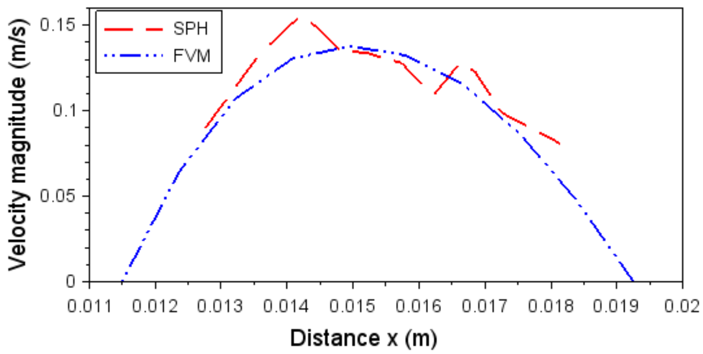

FIGURE 8. Velocity profiles between two blades.

As it was mentioned before, there were tensile instabilities in benchmark case. These instabilities were also observed in case of air diffuser. However only at the beginning of the simulation and places without particles quickly disappeared after the creation. This phenomenon is show on Fig. 9.

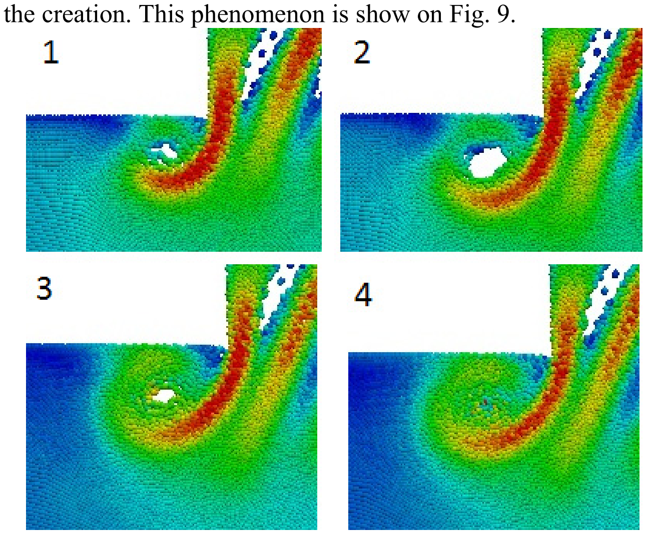

FIGURE 9. Tensile instabilities in case of air diffuser.

CONCLUSION

It has been proved, that SPH is capable for the problem a flow around bluff bodies. Furthermore, the tensile instabilities were identified in case with higher velocity. These instabilities were caused by inappropriate boundary conditions for wall in program DualSPHysics. There is a possible to eliminate these instabilities using smaller particles, which causes an increase of particles in domain and time of computation. There was a good agreement in the velocity profile between results from the FVM and the SPH simulation for both cases. The benefit of SPH is in huge parallelization, which enables computing on graphic card and the speedup is enormous in comparing with computing on CPU. The possibility to use the GPU is now studying for the indoor air flow.

ACKNOWLEDGMENTS

This publication was written at the Technical University of Liberec as part of the project "Experimental and numerical investigation in applied fluid mechanics and energy devices" with the support of the Specific University Research Grant No. 21124, as provided by the Ministry of Education, Youth and Sports of the Czech Republic in the year 2016.

REFERENCES

1. J. J. Monaghan, *Smoothed Particle Hydrodynamic.* Annu. Rev. Astron. Astrophy **30**, 543–574 (1992).
2. B. D. Rogers, R. A. Dalrymple, P. K. Standsby, *Simulation of caisson breakwater movement using SPH.* Journal of Hydraulic Research **48**, 135–141 (2010).
3. J. J. Monaghan, *Simulating free surface flowswith SPH.* Journal of Computational Physics **110**, 399–406 (1994).
4. R. A. Dalrymple, B. D. Rogers, *Numerical modeling of water waves with the SPH method.* Coastal Engineering **53**, 141–147 (2006).
5. A. Skillen, A. Lind, P. K. Stansby, B. D. Rogers, *Incompressible smoothed particle hydrodynamics (SPH) with reduce temporal noise and generalized FIckian smoothing applied to body-water slam and efficient wave-body interaction.* Comput. Methods Appl. Mech. Engrg. **256**, 163–173 (2013).
6. A. Colagrossi, M. Landrini, *Numerical simulation of interfacial flows by smoothed particle hydrodynamics.* Journal of Computational Physics **191**, 448–475 (2003).
7. J. J. Monaghan, A. Kocharyan, *SPH simulation of multi-phase flow.* Comput. Phys. Commun. **87**, 225–235 (1995).
8. A. Mokos, B. D. Rogers, P. K. Stansby, J. M. Dominguez, *Multi-phase SPH modeling of violent hydrodynamics on GPUs.* Comput. Phys. Commun. **196**, 304–316 (2015).
9. S. Suprijadi, F. Faizal, R. R. Septiawan, *Computational Study on Melting Process Using Smoothed Particle Hydrodynamics.* Journal of Modern Physics **5**, 112–116 (2015).
10. A. M. Tartakovsky, P. Meakin, T. D. Scheibe, R. M. Eichler West, *Simulations of reactive transport and precipitation with smoothed particle hydrodynamics.* Journal of Computation Physics **222**, 654-672 (2007).
11. R. Vacondio, B. D. Rogers, P. K. Standsby, P. Mignosa, J. Feldman, *Variable resolution for SPH: A dynami particle coalescing and splitting scheme.* Comput. Methods Appl. Mech. Engrg. **256**, 132-148 (2013).
12. A. J. Crespo, J. M. Dominguez, B. D. Rogers, M. Gomez-Gesteira, S. Longshaw, R. Canelas, R. Vacondio, A. Barreiro, O. Garcia-Feal, *DualSPHysics: Open-source parallel CFD solver based on Smoothed Particle Hydrodynamics (SPH).* Comput. Phys. Commun. **187**, 204–216 (2015).
13. F. Macia, A. Souto-Iglesias, M Antuono, A Colagrossi, *Benefits of using a Wendland kernel for free-surface flows.*6th international SPHERIC workshop, Hamburg, Germany (2011).