MODELLING A PROPELLER IN VARIOUS LIQUIDS WITH CFD-DEM COUPLED SIMULATION METHOD

Ádám Faller[1], Márton Tóth[2], Dániel Horváth[3], Kornél Tamás[4]

[1] BSc student, Budapest University of Technology and Economics, Faculty of Mechanical Engineering, Department of Manufacturing Science and Engineering
H-1111 Budapest, Műegyetem rkp. 3.
fllradam@gmail.hu

[2] MSc student, Budapest University of Technology and Economics, Faculty of Mechanical Engineering, Department of Manufacturing Science and Engineering
H-1111 Budapest, Műegyetem rkp. 3.
t.marton95@gmail.com

[3] PhD student, Budapest University of Technology and Economics, Faculty of Mechanical Engineering, Department of Building Services and Process Engineering
H-1111 Budapest, Műegyetem rkp. 3., Building D 110.
horvath.daniel@mail.bme.hu

[4] PhD, assistant professor, Budapest University of Technology and Economics, Faculty of Mechanical Engineering, Department of Machine and Product Design
H-1111 Budapest, Műegyetem rkp. 3. Building MG 300.
tamas.kornel@gt3.bme.hu

Keywords CFD-DEM, MFiX, propeller, solid impurities, viscosity

Abstract This study is about a task of the BME Solar Boat Team experienced in the Netherlands. The team develops and races an electric racing boat. Due to the low water level in the narrow Dutch channels, the massive upsurge of the sludge can be observed due to the movement of the propeller. After the team started to develop their own propeller, a theoretical approach to the task was also raised. Thus the aim of the interaction between the propeller and the impurities in the water, the dependence of the flow of the profiles and the particles, and the failure of the faster wear, were intended.

In the research, a coupled CFD-DEM simulation method was used, which proved to be appropriate for the task. To construct and execute simulations, the MFiX open source software was used. The ability of the method is demonstrated by the fact that complicated simulations can be simulated with low computing capacity and the fact that there have been several successful simulations for the used method. One of the test aspects was the change in the viscosity of the medium flowing in the chosen geometry, which resulted in different deposition and flows images. As an outline, the method is applicable for various engineering-tribological tasks, which greatly assist the work of design engineers, saving time and money by replacing laboratory measurements with simulations. In addition, they provide guidance on selecting the appropriate geometry or viscosity flow medium for machine elements. The method chosen has proven to be appropriate for examining the problems that arise.

1 INTRODUCTION

The TU Budapest Solar Boat Team met with the premature wear of the propeller first time on their travel to the Netherlands. The reason for this rapid failure is that the channels in the Netherlands which are used for waterborne traffic are very shallow, with the maximum water penetration of the propellers often being regulated. Due to the low water level, the propeller is close to the muddy soil and the currents generated disturb the substrate. The circulating particles in the water have an abrasive effect on the leaves of the propeller. The result of the prolonged wear process changes the propellers attributions, which are no longer able to produce an ideal pressure difference to ensure the movement of the boats.

Figure 1: Solar powered boats in the canals of the Netherlands.

The effect of particle wear often needs to be considered as a design aspect and has to be taken into account when designing the construction. The study of abrasion phenomena often requires expensive and time-consuming experiments. Extensive studies of different geometries are needed to choose the right shape of the propellers. The time, money and alternative costs of this test process can be very high, especially if it is considered the time of a test and the cost of prototyping. Numerical modelling and simulations are the focus of such considerations, which can replace costly tests, the workflow of the designers fastens up, furthermore accurate data can be retrieved from the process, often with information that would not have accessed by conventional experimentations and measurements.

Numerical methods are already widely used to support design. One of the most common of these is the CFD simulation method, which can be successfully used for solving aeronautical problems. In these cases, both the wing profile and any propeller, turbine blade, and airfield interaction should be tested with the particulate medium [1].

Another widespread simulation approach is the discrete element method (DEM). This method is widely used in industrial and agricultural applications in areas where the behaviour of granular materials have to be investigated. Appropriate parameters can effectively simulate the relationship between agricultural machinery and soil [2],[3]. The method also provides outstanding accuracy and results in mixing, transporting and feeding grains [4]. The disadvantage is that there is no interaction between the mediums as fluids containing the granules and the particles so as a result, most processes cannot be simulated with sufficient accuracy.

The coupling of these two modes results in a so-called CFD-DEM, which is suitable for modelling a liquid-particle mixed phase. The method shows promising results for different mixing tasks [5],[6], in agriculture [3] and in more additional areas [7],[8].

The aim of this research was to set up a model containing a propeller, particles and liquids with appropriate boundary conditions and settings for future research. With the improved CFD-DEM model parameter sensitivity tests were performed to investigate the possibilities of reducing abrasive effects.

2 MATERIAL AND METHOD

In this research, simulations were performed by using the MFiX [9] CFD-DEM software, in which the relationship between fluid contaminated with solid particles and a propeller was investigated.

2.1 CFD-DEM METHOD

The coupled CFD-DEM method can be used to model the liquid-particle mixed phase. The liquid part is considered to be a mesoscale phase described by the volume-averaged Navier-Stokes equation [9]. The behaviour of microscale solid particles is described in Newton's and Euler's second law [9]. The body is subjected to gravitational, collision, fluid-particle, particle-particle forces. Usually thousands or even millions of particles are present at the same time in the modelled system, and the equations for them must be solved along with fluid equations [9].

In CFD-DEM modelling, the liquid-particle relationship to the fluid is a function of the number of cell-to-cell interactions, whereas for particles it is determined individually. When the coupling takes place, the position and speed of the particles are calculated by the DEM solver at each timestep, while the CFD solver provides information of the fluid flow profile for the next step in the cell. The resulting profiles can be used to calculate the interaction between fluid and particles.

The coupling condition assumes a completely explicit relationship, which means that the switching conditions are calculated with the information given in the current time step. Then the movement equations of both steps are solved at the same time so that the iterating calculations are stepped forward by one timestep [9].

2.2 SIMULATION ENVIRONMENT

The investigated propeller blade profiles are used by TU Budapest Solar Boat Team. Minor changes were made to these standard NACA wing profiles [11]. The derivation of the profiles is shown on Figure 2. The propeller is cut by a uniaxial cylinder, then the cylinder wall with the projected profile is stretched out. The length of the profile was 121 mm, and the size of the span was practically irrelevant due to 2D simulations.

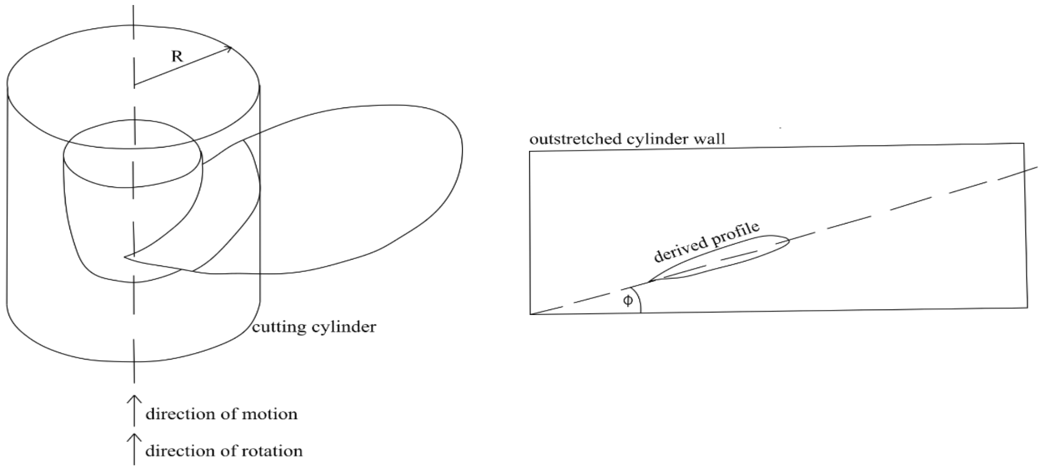

Figure 2: The derivation of the profile. R is the radius belonging to the derived profile. ϕ is the pitch of the derived profile [10].

The CAD model in the simulations was a few centimetres in size. In the program, a flow domain could be determined around the test body (wing profile), which was a rectangle of 0.6 × 0.34 m in subsequent runs (Figure 3). Bearing in mind the constraints of computational capacity and computational accuracy, the domain (computation range) had a resolution of 80 × 40 cells, and a z-direction had been determined for 2D flow. The geometry profiles were defined by splines. The two profiles only differed slightly, and given the nature of their definition exact differences could not be given or shown.

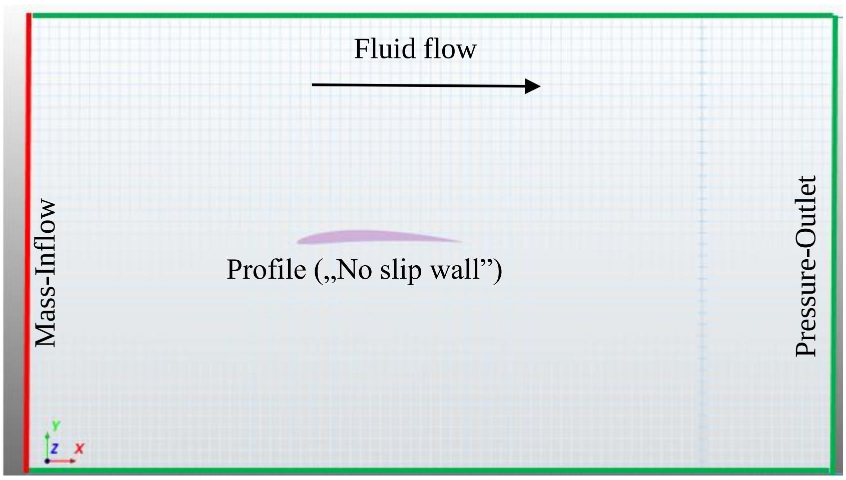

Figure 3: Computational area, Red zone: mass inflow. Green zone: pressure- and material-outlet. The dimensions of the domain are 0.6 m by 0.34 m.

The drag of the particle was according to the MFiX default, with two respective parameters (C_1, D_1) provided by the Syamlal-O'Brien model [12]. Any other parameter was left at its default value.

2.3 FLUID PARAMETERS

In MFiX, many material properties can be specified for inert media, to be gas or liquid, thus, the density of the medium, which in the first case was water, could be also specified, its density was $\rho = 1000$ kg m^{-3}, its molweight was $M = 18.015$ and its dynamic viscosity was $\mu = 10^{-3}$ Pa s. Any other material parameter was set as it is at 20 °C water. Equation solving for each impulse equation could be set, which were only allowed in the x- and y-direction for fluid movement due to the 2D flow.

2.4 PARTICLE PARAMETERS

In MFiX DEM simulations, the dispersed solid phase was modelled with spherical particles. The average diameter of these particles could be approximated by 1 mm in uniform distribution. The density of particles' material was 2600 kg m^{-3}, specified by literature [3-13] basis. For modeling the collision of the particles, a linear spring-dashpot system was selected, which was also used within the literature [3-13]. The micromechanical parameters of the particles can be seen in Table 1.

Table 1: Particle parameters.

Diameter	$D_s = 1\ mm$	
Particle density	$\rho_s = 2600\ kg\ m^{-3}$	
Integral method	Euler-method	
Contact model	Linear spring-damping	
Interpolation	From fluid cell to particle and from particle to fluid zone	
	Particle-Particle	Particle-Wall
Friction coefficient	0.6	0.5
Normal stiffness	1000 N m^{-1}	1000 N m^{-1}
Spring normal/tangent ratio	0.5	0.5
Damping normal/tangent ratio	1	1

3 RESULTS

In the case of the water medium, the result of the first simulation is shown in Figure 4. It is noticeable that in the front part of the wing, about one-fifth of the length of the string, the particles showed congestion. At the same time, most of the pressed side of the wing (upper

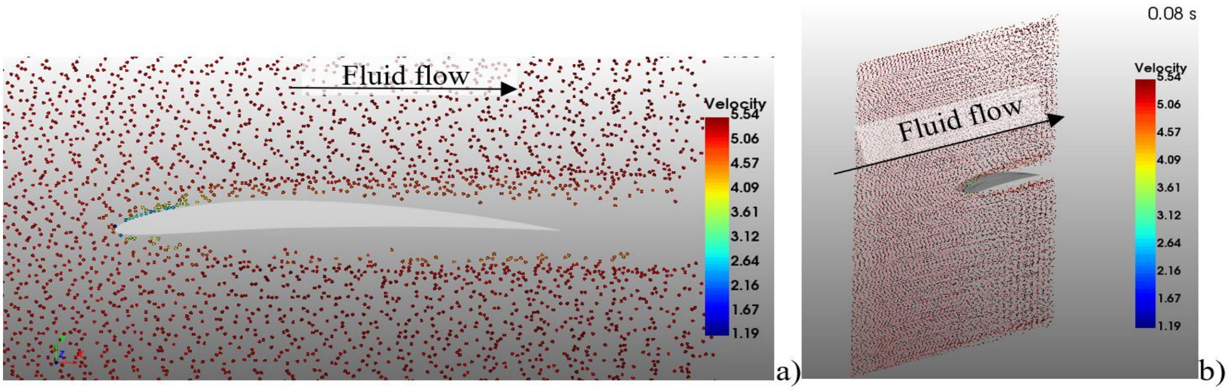

Figure 4: Flow around wing profile in particle contaminated media a) side view b) ortogonal view. Velocity is given in m s^{-1}.

part) did not rub with the particles, and the lower suction side was almost not affected by the particles. Figure 4 shows that the velocity of the particles in the vicinity of the rest of the flow was reduced to 1-2 m s^{-1} compared to the other regions of the wing, where the particles maintained their near-boundary-set speed (5.5 m s^{-1}). It is important to note, however, that this speed is the translational speed. If the angular velocity is displayed, it can be seen the angular velocity of the particles was continuously increased, thus the movement of the particles was only significant after the detachment from the wing, so there is only dealt with the examination of the translational speed. The model of simulation did not include turbulence model which greatly simplified the simulations. The simulation described above was repeated in other media with different viscosities and densities. Based on the results, only qualitative conclusions were drawn from the observation of the obtained figures for the time being. After simulations with some of more well-known media, it could be established that, above a certain viscosity, considering same particle size, the particles covered up the profile. In the case of a viscous substance such as honey, this phenomenon was clear. The process also occurred with materials with lower viscosity, although more research was needed in this field. Additional simulations were run, leaving other parameters unchanged. There was selected materials with different viscosities for the simulations: oil, fruit concentrate and honey, in series $\mu = 0.0006, \mu = 0.007$ and $\mu = 10\ Pa\ s$ viscosities. According to the observations, the critical phenomenon occurred at $\mu = 0.01 - 0.015\ Pa\ s$ which can be seen in Figure 5.

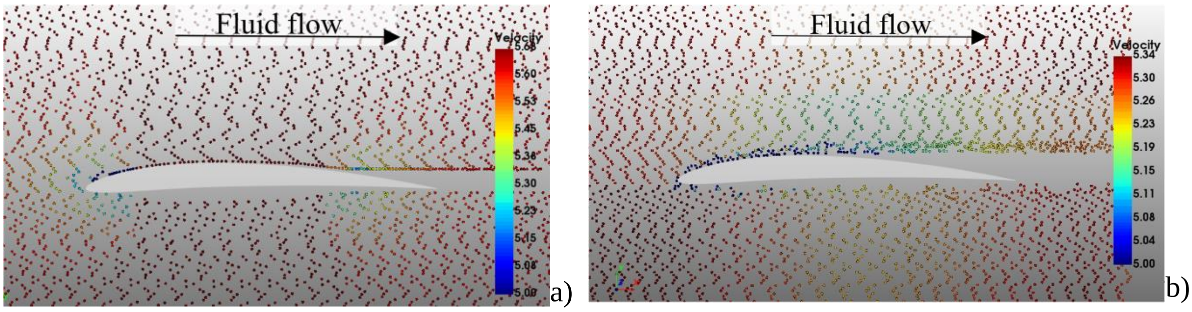

Figure 5: a) Flow around profile in μ= 0.01 $Pa\ s$ viscosity media, translational velocity of particles are displayed, b) Flow around profile in μ= 10 $Pa\ s$ viscosity media, translational velocity of particles are displayed. Velocity is given in m s^{-1}.

4 CONCLUSIONS

The aims of this study were achieved. It has been successfully set up a model for simulating mixed liquid-solid mixed flow media. Parameter sensitivity test was performed. Different qualitative results were obtained for different parameter values. While examining the wing profiles, two types were tested during the simulations, but in the contaminated medium, there was practically experienced the same current image, because these profiles differed only slightly, thus in this paper only one was presented. The improved simulation model included a number of simplifications, as neglect of turbulence and full rotation of the blades, as the flow around the geometry could not be modelled by MFiX. However, it was possible to set a number of material characteristics such as particle density and viscosity, from which qualitative information was obtained about the flow processes in the contaminated environment. However, in order to obtain quantitative, numerical data to investigate the abrasion phenomenon, further study of the software and method is required.

5 ACKNOWLEDGEMENTS

The publication of the work reported herein has been supported by ETDB at BME. The research reported in this paper was supported by the Higher Education Excellence Program of the Ministry of Human Capacities in the frame of Artificial intelligence research area of Budapest University of Technology and Economics (BME FIKP-MI) and the Richter Gedeon Talentum Foundation (19-21. Gyömrői street, 1103 Budapest. Hungary).

REFERENCES

[1] Abras, J., & Narducci, R. (2010). Analysis of CFD modeling techniques over the mv-22 tiltrotor. In American Helicopter Society 66th Annual Forum pp. 11-13.

[2] Li, B., Chen, Y., & Chen, J. (2016). Modeling of soil–claw interaction using the discrete element method (DEM). Soil and Tillage Research, 158, 177-185.

[3] Tamas, K. (2018). The role of bond and damping in the discrete element model of soil-sweep interaction. Biosystems Engineering, 169, 57-70.

[4] Cleary, P. W., & Sawley, M. L. (2002). DEM modelling of industrial granular flows: 3D case studies and the effect of particle shape on hopper discharge. Applied Mathematical Modelling, 26(2), 89-111.

[5] David L.Youngs, (1984). Numerical simulation of turbulent mixing by Rayleigh-Taylor instability, Physica D: *Nonlinear Phenomena* Volume 12, Issues 1–3, July, Pages 32-44.

[6] Deen, N.G., van Sint Annaland, M., and Kuipers, J.A.M. (2004). Multi-scale modeling of dispersed gas–liquidn two-phase flow. *Chemical Engineering Science*, 59(8–9), 1853–1861.

[7] KATHURIA, G., KARNIK, A., & TANDON, M. (2015). STUDY OF GAS-SOLID FLUIDIZED BED-TUNING SYAMLAL O'BRIEN DRAG MODEL. technical report from CD-adapco, DOI: 10.13140/RG. 2.1. 4300.5609/1.

[8] Ge, W., & Li, J. (2003). Simulation of particle–fluid systems with macro-scale pseudo-particle modeling. Powder Technology, 137(1-2), 99-108.

[9] Garg, R., Galvin, J., Li, T., & Pannala, S. (2010). Documentation of open-source MFIX–DEM software for gas–solids flows. From URL https://mfix. netl. doe. gov/documentation/dem_doc_2012-1. pdf.(Accessed: 31 March 2014).

[10] Zoltán Benedek: Ships II. Széchenyi István Technical University, Department of Ships, manuscript (1992)

[11] http://airfoiltools.com/airfoil/naca4digit

[12] https://mfix.netl.doe.gov/mfix/

[13] Horabik, J., & Molenda, M. (2016). Parameters and contact models for DEM simulations of agricultural granular materials: A review. Biosystems Engineering, 147, 206-225.