

ANNA STORY\*, ZDZISŁAW JAWORSKI\*

## EFFECT OF THE NUMERICAL GRID DENSITY ON THE MODELLING OF FLUID FLOW IN A STIRRED TANK WITH A PMT IMPELLER

### WPLYW GĘSTOŚCI SIATKI NUMERYCZNEJ NA MODELOWANIE POLA PRĘDKOŚCI CIECZY W MIESZALNIKU Z MIESZADŁEM PMT

#### Abstract

The paper presents the results of numerical simulations performed for a stirred tank equipped with a PMT type impeller, filled up with a Newtonian fluid. The effects of the grid density and mesh quality and also of the simulation mode on the modelling of fluid flow in a stirred tank were studied. The results are compared with literature data obtained from LDA measurements. It was found that denser numerical grids give more detailed information about generated flow field near the impeller blades. Additionally, better compatibility of predicting and experimental results was obtained in the case of the transient mode simulation, what also demonstrates a significant effect of the angular position of the impeller against baffles on the generated velocity field.

*Keywords: CFD, PMT impeller, velocity field, grid density, mesh quality*

#### Streszczenie

W artykule przedstawiono wyniki symulacji numerycznych prowadzonych dla mieszalnika z mieszadłem typu PMT, wypełnionego cieczą newtonowską. Zbadano wpływ gęstości i jakości siatki numerycznej oraz trybu prowadzenia symulacji numerycznych na modelowanie pola prędkości cieczy w mieszalniku. Wyniki porównano z literaturowymi danymi eksperymentalnymi z pomiarów LDA. Stwierdzono, że gęstsza siatka numeryczna daje bardziej szczegółowe informacje o generowanym polu prędkości w pobliżu łopatek mieszadła. Dodatkowo lepszą zgodność wyników przewidywanych z doświadczalnymi otrzymano w przypadku prowadzenia symulacji w trybie nieustalonym, co świadczy o dużym wpływie kąтового położenia mieszadła względem przegród na generowane pole prędkości.

*Słowa kluczowe: CFD, mieszadło PMT, pole przepływu, gęstość siatki, jakość siatki*

\* M.Sc. Anna Story, Prof. Zdzisław Jaworski, Institute of Chemical Engineering and Environmental Protection Processes, Faculty of Chemical Technology and Engineering, West Pomeranian University of Technology, Szczecin.

## 1. Introduction

Computational Fluid Dynamics (CFD) uses numerical software in a way to get the best possible solution of the partial differential equations of transport processes in fluids. It must be mentioned, that the numerical analysis, which is conducted to obtain the correct solution is a multi-stage and difficult process, and the transition from the first stage, which consists of a preliminary problem analysis, to the last – results analysis, requires application by an operator specialized and advanced knowledge about the transport process modelling and numerical methods [1]. Here, a very important stage is to select an appropriate numerical grid, which should satisfy two requirements – faithfully reproduce the modelled volume of fluid and properly divide the volume into computational cells. At present, automatic mesh generators are available (e.g. Gambit, Workbench Meshing, Tgrid), and for this reason the main problem is concerned with i) selection of elements of which the numerical mesh will be made (tetrahedral, hexahedral) and ii) choice of the mesh density, which directly affects the solution accuracy. Basically, there are two main types of numerical grids: structural and non-structural [1]. The structural grids are whole constructed of hexahedral elements and their surfaces have four edges. They are characterized by a regular structure, which limits their applicability to a relatively simple geometry and for more complex geometries the block-structural grids can be used. When using the structural type of mesh, the modelled volume is divided into blocks, in which different structural meshes are generated. The non-structural grids can be constructed of elements of any shape, but the most commonly used are tetrahedral, pentahedral and hexahedral elements. Different shapes of the non-structural grid elements allows the use of this type of mesh for modelling the volumes of very complex geometries.

It has been frequently proven that a denser mesh gives more accurate numerical solutions of the analysed problems [2–6]. For example Peryt-Stawiarska et al [5] analysed the effect of the numerical grid density on the modelled apparent viscosity and hydrodynamics of non-Newtonian fluid flows in a stirred tank equipped with the Rushton turbine. They found that a slightly better compatibility of numerical simulations results with corresponding experimental data was obtained for a denser numerical grid. Karcz and Kacperski [6] performed numerical simulations of Newtonian fluid flow in a mixing tank equipped with a 6 pitched blade turbine impeller for six numerical grids of different density. The simulation results showed that numerical grid density substantially influences the distribution of kinetic energy and its dissipation. The velocity fields were similar and the differences can be seen very close to the impeller blades only. The authors concluded that an increase of numerical cell number gives a more detailed maps of the local flow characteristics. On the other hand, the use of denser numerical grids is associated with a significant increase of the and required computing power needed to perform numerical simulations.

The purpose of this paper is to analyse the effect of the numerical grid density on the results of numerical simulations of the Newtonian fluid turbulent flow in a mixing tank equipped with a PMT type impeller, which has geometry similar to the Prochem Maxflo T impeller. The data obtained from the CFD numerical modelling were compared with experimental data available in the literature [7] obtained by Laser Doppler Anemometry (LDA).

## 2. Experimental

### 2.1. The studied system

The numerical simulations were performed for a flat-bottomed stirred tank of a diameter  $T = 0.222$  [m]. The tank was filled up with a Newtonian liquid (water) to the height  $H = T$ . The stirred tank was equipped with four flat baffles with a width  $B = 0.1T$  and a centrally located PMT type impeller with a diameter  $D = 0.35T$ . The center of the agitator was in a distance of  $z = 0.1$  [m] from the tank bottom. The agitator rotated at a constant frequency of  $N = 4.1$  [ $s^{-1}$ ]. Geometry of the system and the process parameters were the same as those used in LDA measurements described in the literature [7].

In the ANSYS Workbench Meshing 14.5 preprocessor, three numerical grids with different number of computational cells were generated. The first grid divided the volume of the stirred liquid into about 79 thousand, the second into about 798 thousand and the third into about 2 million computational cells (Fig. 1). The average values of the basic parameters defining the quality of the generated numerical grids are summarized in Table 1.

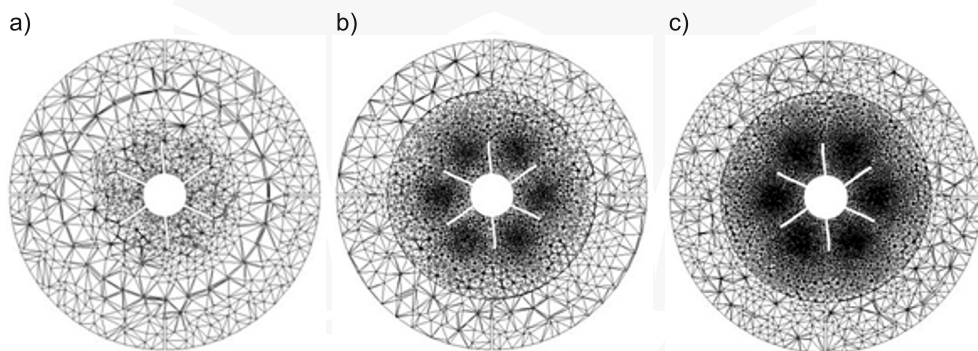


Fig. 1. Generated numerical grids in a horizontal plane located at the axial position  $z = 0.1$  [m] from the tank bottom: a) 79 251 cells, b) 798 237 cells, c) 2 001 880 cells

Table 1

Parameters of the generated numerical grids

	Grid 1	Grid 2	Grid 3
Numerical cells number	79 251	798 237	2 001 880
Element Quality	0.77	0.84	0.84
Skewness	0.34	0.23	0.22
Orthogonal Quality	0.81	0.86	0.86

Analyzing the parameters of the generated numerical grids it was found, that grid 1 (79 thousand computational cells) was characterized by the worst quality. For this reason it is assumed, that the worst results of numerical simulations will be obtained for this grid. In the case of denser grids (798 thousand and 2 million of computational cells), their quality

was similar, but slightly better was the mesh with a density of 2 million computational cells. The increase of the computational cells number caused about 6-times and about 23-times increase of calculation time respectively for grid 2 and 3, compared to the calculation carried out for the first grid.

## 2.2. Numerical simulations

Numerical simulations were carried out for the turbulent flow of the modelled fluid, the calculated value of Reynolds number for mixing was  $Re = 24\,800$  [–]. The commercial CFD software ANSYS Fluent 14.5. was used. Modelling was of a three-stage character. In the first step simulations were performed for the laminar flow of water using a Multiple Reference Frame (MRF) method. After reaching convergence of the iterations, the MRF simulations were continued with enabled turbulence model. To simulate the turbulent flow the Reynolds-averaged Navier-Stokes equations (RANS) method with the  $k$ - $\varepsilon$  turbulence model (1a, b) [1] and the standard wall functions were used.

$$\frac{\partial k}{\partial t} + \text{div}(k\bar{\mathbf{v}}) = \text{div}\left(\frac{v_t}{\sigma_k} \text{grad } k\right) + P - \varepsilon \quad (1a)$$

$$\frac{\partial \varepsilon}{\partial t} + \text{div}(\varepsilon\bar{\mathbf{v}}) = \text{div}\left(\frac{v_t}{\sigma_\varepsilon} \text{grad } \varepsilon\right) + C_{\varepsilon_1} \frac{P\varepsilon}{k} - C_{\varepsilon_2} \frac{\varepsilon^2}{k} \quad (1b)$$

The turbulent viscosity of Equations (1a, 1b) is computed from:

$$v_t = C_\mu \frac{k^2}{\varepsilon} \quad (2)$$

and the turbulent Prandtl numbers  $\sigma_k$ ,  $\sigma_\varepsilon$  and coefficients  $C_{\varepsilon_1}$ ,  $C_{\varepsilon_2}$  are the empirical constants of the model.

After completion of the second stage of the simulation, the grid 2 was selected. For this grid the numerical simulation was continued in the transient mode using the Sliding Mesh (SM) method with a small time step. The time step corresponded to the rotation of the impeller for an angle of  $2^\circ$ . For that time step the calculated value of the convective Courant number ( $C_k = v\Delta t/\Delta x$ ) was equal to 1, assuming that the fluid velocity,  $v$ , is equal to the peripheral velocity of the blades ( $v_{\text{TIP}} = \pi DN$ ). The peripheral velocity of the impeller is the maximum velocity that can be reached by the liquid. In practice, the velocity of the liquid attained values lower than  $v_{\text{TIP}}$ , thus the value of  $C_k$  was then less than 1. Thus, the condition, which provides stability of the calculation,  $C_k < 1$  [1], was met. The number of internal iterations in one time step was equal to 100. The simulations in the transient mode, by changing the angular position of the impeller in time, reflect the effect of agitator position relative to baffles on the generated velocity field. The final results of the numerical simulations were read for a vertical measuring plane located at the angle of  $45^\circ$  between the baffles.

### 3. Results and discussion

Maps of the mean velocity obtained after the completion of the second stage of numerical simulations conducted by MRF method are presented in Figure 2, for the three grids of different number of computational cells.

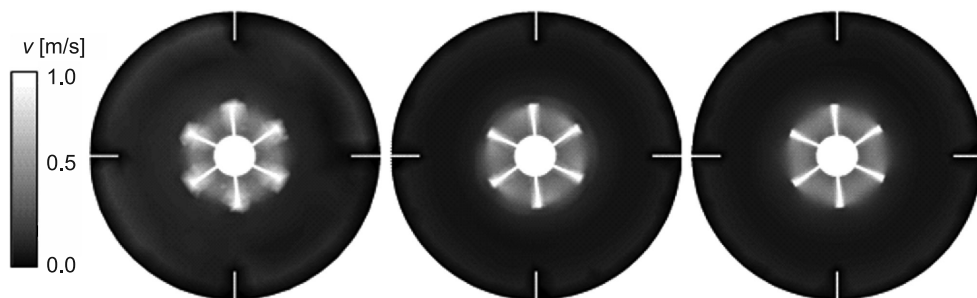


Fig. 2. Results of MRF numerical simulations with enabled  $k-\varepsilon$  turbulence model in a horizontal plane located at the axial position  $z = 0.1$  [m] from the tank bottom: a) 79 251 cells, b) 798 237 cells, c) 2 001 880 cells

Based on the average velocity distributions with their maps presented in Fig. 2, it was found that qualitative and quantitative distribution of the fluid velocity in the mixing tank was similar for all the numerical grids. The differences can only be observed in the velocity distribution close to the impeller blade region. These differences are especially apparent in the case of the lowest grid density and the worst quality mesh. There were no significant differences between the distribution of velocity in the case of the meshes with densities 798 thousand and 2 million computational cells. Therefore, because of the much shorter calculation time for the grid with 798 thousand cells, this grid was selected to further simulation performed in the transient mode.

After completion of calculation in the transient mode the mean velocity vectors in a mid-plane between two neighbouring baffles were computed from results of the numerical simulations and presented in Figure 3. The velocity vectors were compared with analogous results obtained from the published LDA measurements [7]. Analyzing the predicted distribution of the mean velocity vectors shown in Fig. 3a it was concluded, that their visual distribution was similar to those obtained from experimental measurements (Fig. 3b). There were one primary and two secondary circulation loops in the flow, and the PMT type impeller created a typical axial-radial circulation. The vectors of mean velocity obtained from CFD simulations indicate a good agreement of the predicted and experimental results. However, in order to quantitatively verify the obtained results a numerical analysis was also performed.

For this purpose, the simulated values of three components of instantaneous velocity,  $v_{i,CFD}$  ( $i = z, r, t$ ), from ninety consecutive time steps were collected, what corresponded to the rotation of the impeller by the angle of  $180^\circ$ . The values of the local instantaneous velocity were next averaged and standardized by the peripheral velocity of the impeller,  $v_{TIP}$ , giving a dimensionless values of the average velocity,  $\bar{V}_{I,CFD}(I = Z, R, T)$ . Next, the profiles

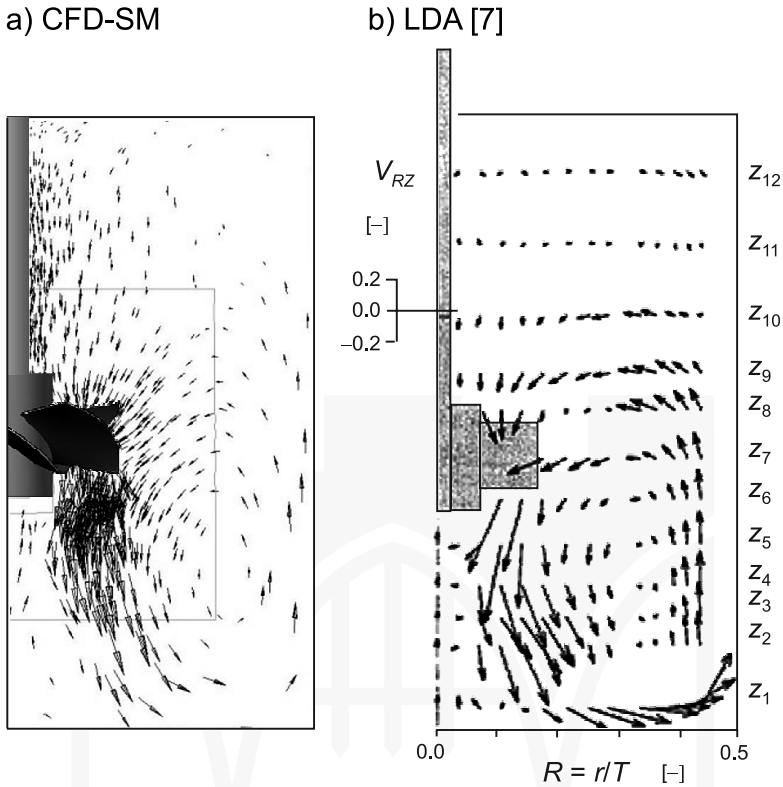


Fig. 3. Vectors of mean velocity in a mid-plane between two neighbouring baffles: a) numerical simulations, b) LDA measurements [7]

of three dimensionless components of the mean velocity along the non-dimensional radius ( $R = r/T$ ) at different axial positions were plotted. The positions were chosen in the range of  $z = -80 \div 100$  [mm], with the value  $z = 0$  [mm] being the axial coordinate of the center of the impeller. Sample profiles of the dimensionless average velocity components at height  $z = 17$  [mm] are shown in Figure 4a. Profiles of the three non-dimensional velocity components obtained after the second step of the simulation (using the MRF method) and from the literature data [7] for the LDA measurements carried out for the same system are shown in Fig. 4a. Analysing the mean velocity profiles, which are shown in Figure 4a it was found that velocity distributions predicted by numerical simulations were qualitatively similar to that obtained from experimental LDA measurements.

Based on the instantaneous values of the three components of velocity,  $v_{i,CFD}$ , obtained from the simulations for ninety successive time steps, the values of fluctuating velocity components,  $v'_{i,CFD}$ , in the axial, radial and tangential direct were also calculated. The values of the fluctuating velocity components were divided by the peripheral velocity of the impeller,  $v_{TIP}$ , resulting in a dimensionless values of the fluctuating velocity

components,  $V'_{I,CFD}$ . The profiles of dimensionless values of the fluctuating velocity components were shown in Figure 4b.

Subsequently, the values of the mean square deviations between the mean velocity components obtained from the numerical simulations performed by the MRF and SM methods and those from the literature [7] data for the LDA measurements were calculated from equation (3):

$$\sigma = \sqrt{\frac{\sum (\bar{V}_{I,LDA,i} - \bar{V}_{I,CFD,i})^2}{(n-1)}} \cdot 100\% \quad (i = 1 \dots n) \quad (3)$$

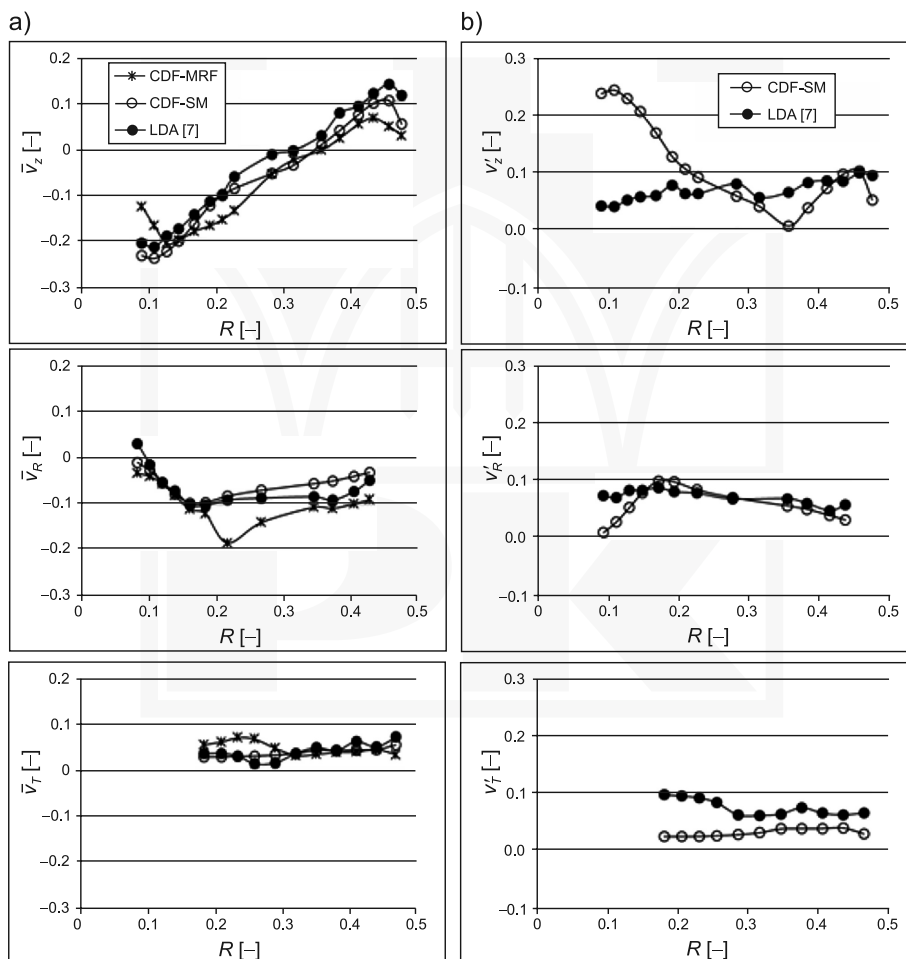


Fig. 4. Dimensionless velocity profiles along the dimensionless radius at the height  $z = 17$  [mm] obtained by two measurement techniques: a) mean velocity, b) fluctuating velocity

The values of the mean square deviations are summarized in Table 2.

Table 2

**Values of the mean square deviations between experimental [7] and predicted values of mean velocity**

	Axial component of mean velocity	Radial component of mean velocity	Tangential component of mean velocity
	Mean square deviations, $\sigma$ [%]		
LDA/CFD-MRF	17.7	8.4	2.4
LDA/CFD-SM	5.0	3.9	1.9

The average values of mean square deviations summarized in Table 2 indicate a better compatibility of the experimental data with the predicted results obtained for the simulation carried out in the transient mode. The highest value of the mean square error for the simulations performed by the SM method was obtained for the axial component of the mean velocity. However, that value did not exceed 5%. In the simulations conducted in the steady state mode for the same component of the mean velocity the value of  $\sigma$  was more than three times higher and equal to 17.7%. The best agreement of the predicted results with experimental ones was obtained for the tangential component of the average velocity. The obtained values of  $\sigma$  for the MRF and SM simulations indicate a large effect of impeller angular position relative to baffles on the generated velocity field.

#### 4. Conclusions

The paper presented numerical simulations results of turbulent flow of a Newtonian fluid. The simulations were performed for the mixing tank equipped with the PMT type impeller. Three numerical grids with different number of computational cells (79 thousand, 798 thousand and 2 million cells) were generated for the test system. It was found that the grid density significantly affects the mesh quality and the modelled velocity field in a mixing tank. The differences can be particularly observed in the distribution of velocity close to the impeller blade region. For denser grids, which are characterized by a better quality, the generated velocity fields were also similar. In addition, the effect of the simulation mode on the predicted velocity field was considered. The results were compared with the literature data from the LDA measurement. A better compatibility of predicted results with the experiment was obtained in the case of the transient mode simulation using the Sliding Mesh method. This demonstrates the significant influence of the angular position of the impeller against baffles on the generated velocity field.

*The authors are grateful to Professor Alvin W. Nienow who agreed to use the original LDA data [7].*



## References

- [1] Jaworski Z., *Numeryczna mechanika płynów w inżynierii chemicznej i procesowej*, Akademicka Oficyna Wydawnicza EXIT, Warszawa 2005.
- [2] Aubin J., Fletcher D.F., Xuereb C., *Modelling turbulent flow In stirred tanks with CFD*, Exp. Therm. Fluid Sci., vol. 28, 2004, 431-445.
- [3] Deglon D.A., Meyer C.J., *CFD modeling of stirred tanks: Numerical considerations*, Miner. Eng., vol. 19, 2006, 1059-1068.
- [4] Zhang Q., Yong Y., Mao Z., Yang C., Zhao C., *Experimental determination and numerical simulation of mixing time in a gas – liquid stirred tank*, Chem. Eng. Sci., vol. 64, 2009, 2926-2933.
- [5] Peryt-Stawiarska S., Zakrzewska B., Jaworski Z., *Modelowanie CFD przepływu przejściowego płynu nienewtonowskiego w mieszalniku z turbiną Rushtona*, Przemysł Chemiczny, vol. 90(9), 2011, 1664-1666.
- [6] Karcz J., Kacperski Ł., *An effect of grid quality on the results of numerical simulations of the fluid flow field in an agitated vessel*, 14<sup>th</sup> European Conference on Mixing, Warszawa, 10–13 September 2012, 205-210.
- [7] Jaworski Z., Nienow A.W., *LDA measurements of flow fields with hydrofoil impellers in fluids with different rheological properties*, IChemE Symp. Series No. 136, 1994, 105-112.

