Contents lists available at ScienceDirect

Chemical Engineering Research and Design

journal homepage: www.elsevier.com/locate/cherd

Large eddy simulation of a pitched blade impeller mixed vessel – Comparison with LDA measurements

Petr Vlček[a,*], *Bohuš Kysela*[b], *Tomáš Jirout*[a], *Ivan Fořt*[a]

[a] *Czech Technical University in Prague, Faculty of Mechanical Engineering, Department of Process Engineering, Technicka 4, 166 07 Prague 6, Czech Republic*
[b] *Institute of Hydrodynamics, AS CR, v.v.i., Pod Patankou 30/5, 16612 Prague 6, Czech Republic*

ARTICLE INFO

Article history:
Received 20 August 2015
Received in revised form 4 December 2015
Accepted 15 February 2016
Available online 24 February 2016

Keywords:
CFD
LES
Mixed vessel
Pitched six-blade impeller
Ensemble averaged mean velocity
Flow rate

ABSTRACT

This study deals with a large eddy simulation (LES) of the single-phase turbulent flow in a fully baffled mixed vessel stirred by a pitched six-blade impeller. The ANSYS FLUENT 15.7 commercial CFD programme was employed. The LES turbulent model is used with the dynamic Smagorinsky–Lilly model. The Werner–Wengle wall function is used to cope with the requirements of near the wall cell size of the LES model. The sliding mesh (SM) model was used for simulating the impeller movement. The results of the simulations are compared with experimental data obtained by previous LDA measurements of the radial profiles of the axial component of the ensemble averaged mean velocity in the impeller discharge flow. The volume flow rates and the flow rate criteria were calculated from these velocity profiles, and were compared with the flow rates determined from the horizontal vessel cross section from simulation.

1. Introduction

The future design and development of agitated tanks, as well as whole engineering, is focused on individual solutions for each solved case; to achieve the best efficiency and to decrease operating costs. Hence, computational fluid dynamics (CFD) calculations are used more often, namely to predict the power consumption of the impeller, pumping capacities, etc. These basic properties of axial pitched blade impellers are also continuously investigated in experimental studies e.g. pumping capacity (Fort et al., 2002), shapes and geometry (Kumaresan and Joshi, 2006), and the use of multiple impellers (Machado et al., 2013). It is equally important to make more accurate estimates of the dissipation rate and of the distribution of turbulent kinetic energy, which is one of the main aims of the experimental research in the long term (Kresta and Wood, 1993a; Bugay et al., 2002; Guida et al., 2010), and is still under investigation using CFD. Investigations of trailing vortices are also being made by Schafer et al. (1998), Ranade et al. (2002) and Roy et al. (2010), based on a estimates of the turbulent kinetic energy distribution and on knowledge of the basic flow around pitched blade impellers. The flow around pitched blade impellers has been investigated primarily with the use of laser Doppler anemometry (LDA) measurements (Ranade and Joshi, 1989; Kresta and Wood, 1993b; Schafer et al., 1998; Jaworski et al., 2001; Wu et al., 2001). Their findings have been used for CFD validations (Tsui et al., 2006; Tyagi et al., 2007; Jahoda et al., 2007). Numerical simulation can thus help with a more detailed description of the flow and the velocity field in a mixed vessel. A review of CFD research on axial impellers was published by Joshi et al. (2011). It was found in previous works dealing with two-equation k–ε turbulence modeling (Bakker et al., 1997; Ranade et al., 2002; Tsui et al., 2006), that the simulation results did not always coincide with experiments. Transient solutions in combination with LES turbulence modeling (e.g. Roussinova et al., 2003; Tyagi et al., 2007; Jahoda et al., 2007; Roy et al., 2010) provided distinctly better results, but fine calculation adjustment and experimental validation are also needed.

* *Corresponding author*. Tel.: +420 732409804.
E-mail address: petr.vlcek@fs.cvut.cz (P. Vlček).
http://dx.doi.org/10.1016/j.cherd.2016.02.020

Symbols

b	baffle width [m]
c	axial distance between the lower edge of the impeller and the measured radial profile [m]
D	vessel diameter [m]
d	impeller diameter [m]
H	height of the static liquid surface above the vessel bottom [m]
H_2	impeller clearance [m]
h	height of the impeller blade [m]
M_k	torque [N m]
N_{Q_i}	induced flow rate criterion [–]
N_{Q_p}	flow rate criterion [–]
N_{Q_t}	total flow rate criterion [–]
n	impeller speed [rpm]
P	power [W]
Po	power number [–]
Q	flow rate [$m^3\,s^{-1}$]
Q_i	induced flow rate [$m^3\,s^{-1}$]
Q_p	impeller pumping capacity [$m^3\,s^{-1}$]
Q_t	total flow rate [$m^3\,s^{-1}$]
R	dimensionless radial coordinate [–]
Re	Reynolds number; $Re=\rho n d^2/\mu$ [–]
r	radial coordinate [m]
t	blade thickness [m]
W_{ax}	dimensionless mean axial velocity [–]
W'_{ax}	dimensionless RMS of fluctuating axial velocity [–]
$\bar{w}_{ax}$	mean axial velocity [$m\,s^{-1}$]
W'_{ax}	fluctuating axial velocity [$m\,s^{-1}$]
y^+	dimensionless wall distance [–]
z	axial coordinate [m]
μ	dynamic viscosity [Pa s]
ρ	density [$kg\,m^{-3}$]

Flow rate criteria N_{Q_p} and total flow rate criteria N_{Q_t} have been measured by many authors using various methods. Table 1 contains the results of several studies using a partly similar geometry to that used in this work.

Our paper deals with an unsteady simulation of the turbulent flow in a baffled vessel mixed by a six-blade impeller with inclined blades, using the LES model. Several simplifications will be used for the simulation. The results will be compared with data measured using the LDA method and from the literature.

2. Problem analysis

The LES (large eddy simulation model) is used for simulating turbulent flow. It is based on the solution of large eddies as spatially and temporally dependent formations that can be captured by a mesh. Because only large scales are resolved, a much coarser mesh and a larger time step can be used in LES than in DNS (direct numerical simulation). However, in comparison with RANS (Reynolds averaged Navier–Stokes) methods, very fine meshes are required, and the calculations are much more computationally expensive. A major disadvantage of LES is the high resolution requirements for near wall cells in all directions ($y^+=1$, $\Delta x^+\approx 20$, $\Delta z^+\approx 20$), because large turbulent scales are geometrically small near the wall. This essentially limits the applicability of the LES model to flow at low Reynolds numbers and small solved area (ANSYS, 2012a). Various RANS/LES hybrid models or wall functions can be used to remove this lack for complicated geometries (ANSYS, 2012a,b).

The sliding mesh (SM) method, which is suitable for time-dependent simulation, will be used to simulate the rotation of the impeller.

Only single-phase flow will be considered, because the vessel contains four radial baffles, so there is no central vortex formation. Waves on the surface will be neglected, and the symmetry boundary condition will be used at the place of surface.

3. Solution

The flow is solved in a vessel with a flat bottom and four baffles mixed by six-blade impeller with inclined blades at an angle of 45°, at a speed of 300 rpm, pumping liquid downwards toward flat bottom of the vessel. The vessel is filled with water.

The geometry for CFD simulation was created using ANSYS DesignModeler. The basic dimensions of the impeller and the vessel are shown in Fig. 1 and in Table 2. The volume of the vessel was divided into a rotating part and a stationary part, according to the requirements of SM method. In order to create partially structured meshes, the models were further decomposed to sub-volumes, see Fig. 1.

The computing meshes were created in the ANSYS Meshing programme. The rotating volume containing the complicated geometry of the impeller was meshed by means of tetrahedrons, and the stationary part was meshed by means of hexahedrons. Several meshes were gradually created, which differed in the size of the cells, and especially in the refinement of the cells near the wall.

It was found that, for the available computation power, such a fine computation mesh cannot be used for of $y^+=1$, because it would extend the simulation time disproportionately. A mesh with 1,033,000 elements was therefore finally used for simulation with a LES turbulent model with a dynamic Smagorinsky–Lilly subgrid model and the Werner–Wengle wall function.

Given that the LES model provides a transient simulation, it is necessary to time-average the obtained values in order to

Table 1 – Selected results of the flow rate criteria for a six-pitched blade impeller presented by various authors.

	Fort et al. (1975)	Medek and Fort (1979)	Ranade and Joshi (1989)	Jaworski et al. (1991)	Kresta and Wood (1993a)	Aubin et al. (2001)	Kumaresan and Joshi (2006)
N_{Q_p}	0.98	0.9937	0.85	0.73	0.829	0.75 (down) 0.68 (up)	1.08
N_{Q_t}	–	–	1.9	1.38	–	1.82 (up)	1.90
H_2/D	1/4	1/4	1/3	1/3	1/4	1/3	1/3
h/d	0.2	0.2	0.2	0.2	0.2	–	0.18
d/D	1/3	1/3	1/3	1/3	1/3	1/2	0.34
Method	Photographic	Flow follower	LDA	LDA	LDA	LDV	LDA