Supplementary material

CFD model setup

Both the system with a Rushton Disc Turbine (RDT) and a Pitch Blade Turbine (PBT) consists of an inner and an outer region. The inner region, containing the impeller and sparger, is configured as a rotating domain with a "Transient Rotor Stator" with automatic pitch change as the interface model. The top of both the inner and outer region is set to "open" to let fluid leave and enter at the surface as expected in the real system.
The boundary conditions of the inner and outer wall, baffles, sparger, and impeller were all set to no-slip smooth wall. The impeller was set as counter-rotating. The setup was done in accordance with similar studies [1].

Simulations

The simulations were performed in three steps. Initially, a steady state simulation was performed to reduce the computational time of the transient simulation. For the transient simulation, the timestep was set to 0.001 seconds, at which the Courant number was lower than 1. This is done to avoid information to be propagated from multiple cells at one timestep. In the third and last step, the flow field was frozen, and a tracer was initialized in the volume. The simulation was prolonged while the tracer concentration was monitored at four heights.

CFD Mesh

The meshes were produced in ANSYS ICEM in accordance with the quality parameters given in Table S1 (ANSYS CFX-Solver Modelling Guide 12.1. Release 12.1, ©2009 ANSYS, Inc).

Table S1. Mesh quality parameters given in the modelling guide from Ansys CFX [2].

Quality parameter	**Acceptable range**
Orthogonal Quality	[1/3;1]
Mesh Expansion Factor	[1;20]
Max ratio	[1;3]
Aspect ratio	[1/3;1]

Mesh convergence

A mesh convergence study was performed to investigate if the result of the CFD model was independent of the mesh. Two different meshes were produced for each type of impeller. The density in elements pr. L were chosen like the once described in [21]. With these two meshes a tracer simulation was performed at the lowest impeller speed (60rpm for RDT and 105rpm for PBT), and the 95% mixing time was found. The lowest impeller speed was chosen since this is where the turbulence condition will be least developed. Like previous studies it is assumed that the mesh is converged if the relative difference in mixing time found from the simulation with high- and the low-density mesh is below 5%.

Table S1. Results of mesh study.

	PBT		RDT	
Density [#/L]	2799	7721	2894	7169
95% Mixing time [s]	27.2	27.8	45.8	45.2
Relative deviation [%]	2.2	-	1.3	-

From Table S2 it is evident that the change in mixing time from the high- to the low-density mesh is below 5% and therefore simulations using the low-density mesh is satisfactory for the scope of the article.

[21] Bach C, Yang J, Larsson H, Stocks SM, Gernaey K V., Albaek MO, et al. Evaluation of mixing and mass transfer in a stirred pilot scale bioreactor utilizing CFD. Chem Eng Sci 2017;171:19–26. https://doi.org/10.1016/j.ces.2017.05.001.