Design of a gas-inducing impeller using Computational Fluid Dynamics

Pereira Rosinha Grundtvig, Ines; Hybschmann, Tim; Gernaey, Krist V.; Svendsen, Tore C.; Krühne, Ulrich

Publication date:
2017

Document Version
Publisher's PDF, also known as Version of record

Design of a gas-inducing impeller using Computational Fluid Dynamics

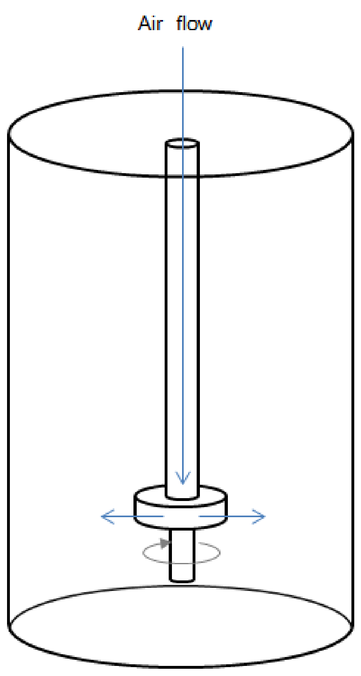
Ines Rosinha Grundtvig[1], Tim Hybschmann[1], Krist Gernaey[1], Tore C. Svendsen[2], Ulrich Krühne[1]

[1]PROSYS Research Center. Department of Chemical and Biochemical Engineering, Technical University of Denmark, 2800 Kgs. Lyngby, Denmark
[2]BIO-AQUA, Strøbjergvej 29, 3600 Frederikssund, Denmark
inros@kt.dtu.dk

Introduction

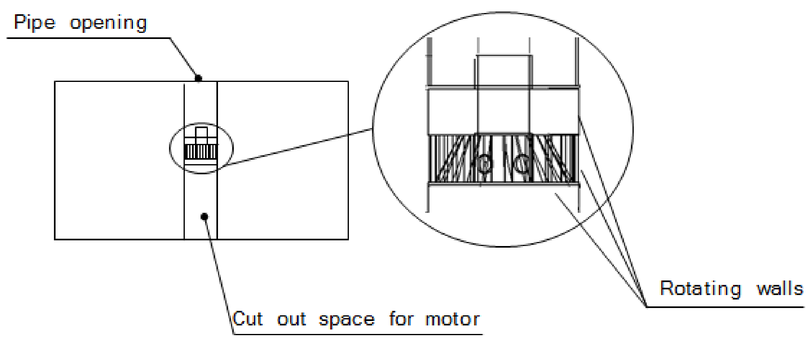
Industrial bioreactors are characterized by non-uniform fluidic conditions due to poor mixing which results in heterogeneous substrate concentration profiles and consequently, low yield reaction systems. The impeller system used is a crucial component to promote the mixing inside bioreactors. This work is an investigation of the design of an impeller of a BIO-AQUA system. BIO-AQUA is a Danish company specialized in designing and developing water solutions for industrial and environmental challenges. The company has developed a fixed film biological reactor for aerobic decomposition of organic matter. The dispersion of air into the system is achieved by using a gas-inducing impeller. The main goal of this computational investigation is to study the impeller design influence on the performance of the system (mass flow sparged into the system and power consumption). Three designs of impellers with different types of blades were chosen for this study: rectangular blades, rectangular blades with round edges and round blades. The computational fluid dynamics (CFD) software, ANSYS CFX® was used as a tool in order to model and analyze the performance of the gas-inducing impeller.

Scheme of a general gas-induced impeller



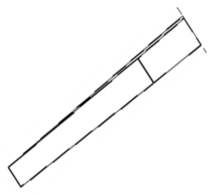
The rotational velocity of the impeller leads to the acceleration of the liquid around it and to a decrease in pressure. The gas-inducing impeller generates a pressure difference in the hollow shaft and is therewith transporting air from the opening at the top of a hollow shaft into the liquid.

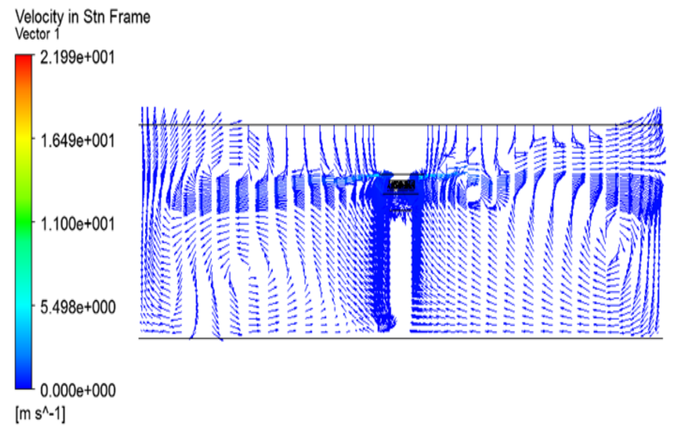
Geometry modelled using Computational Fluid Dynamics



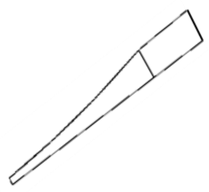
Results

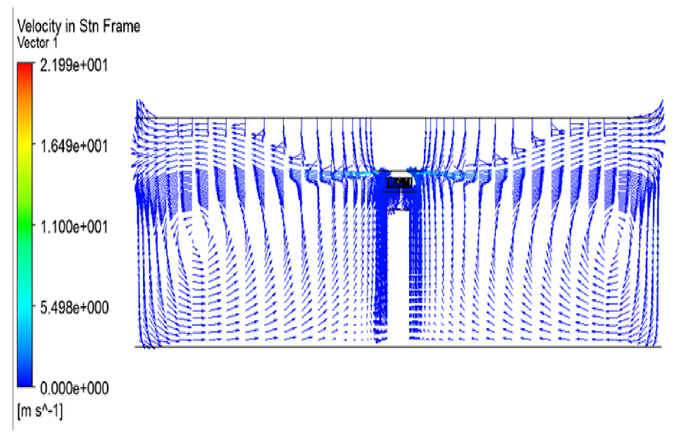
Rectangular blades



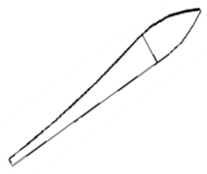


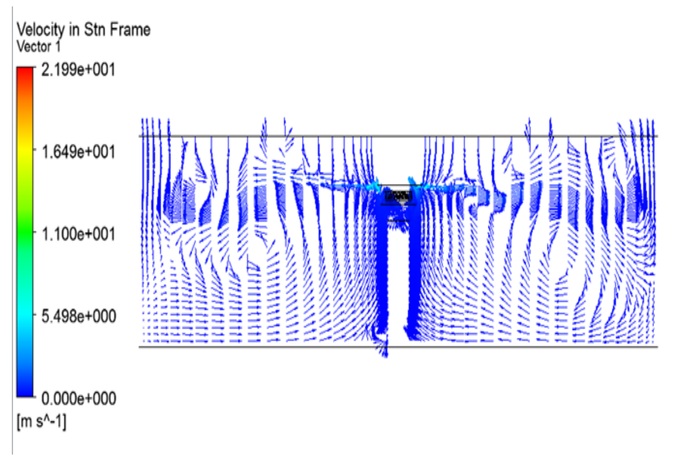
Rectangular blades with round edges



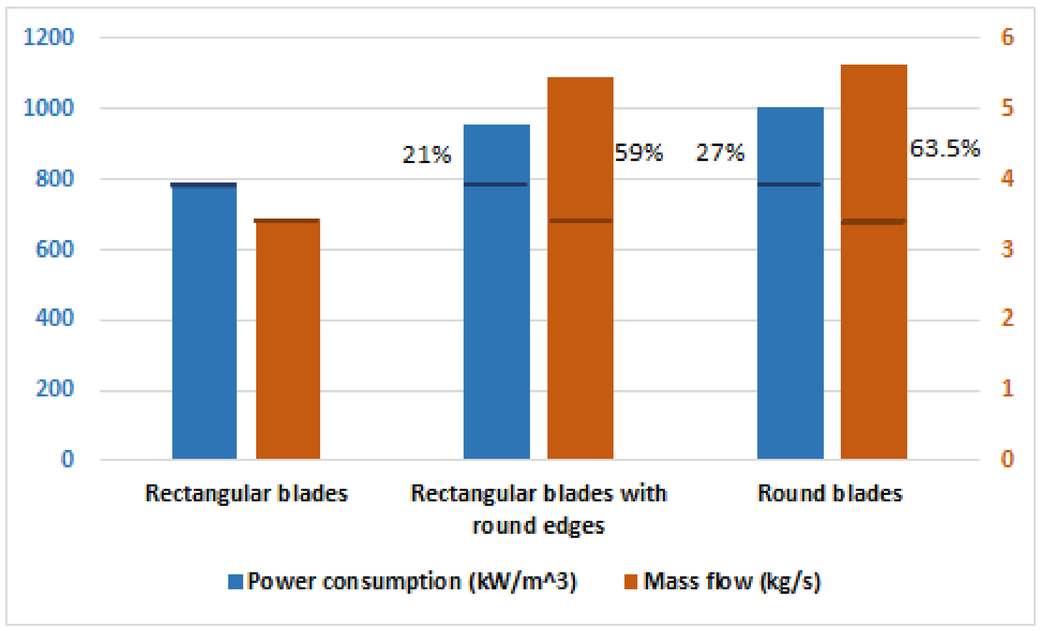


Round blades





Test design results—power consumption and mass flowrate



Conclusions

The results of this test design have shown that the changes made on the blades shape of the impeller have a significant impact on power consumption and on the mass flow sparged into the the system. Even though the blades with more round edges consumed more power than the original design (between 21-27%), they also sparged between 59% and 63,5% more flow into the vessel. The company will make a decision of the type of blades to be used in the system according to the balance between the power consumption and mass flow.

In conclusion, this study has demonstrated how useful CFD is on designing new systems and consequently, minimizing experimental work in order to find best performing system.

References

A.W. Patwardhan, J.B. Joshi, Design of Gas-Inducing Reactors, Industrial and Engineering Chemistry Research, 38 (1999) 49–80

Q. Ye, Z. Li, H. Wu, Principle and performance os gas self-inducing reactors and applications to biotechnology, Advances in Biochemical Engineering / Biotechnolgy, 152 (2016) 1-33

The research leading to these results has received funding from Smart Innovation, Scion DTU. The authors would like also to thank to BIO-AQUA for all the support.

Scion DTU
Science and Technology Park

BIO·AQUA