

REFERENCE LIST

- [1] J. Van Der Linden, “Liquid-liquid separation in disc stack centrifuge,” Master’s thesis, Delft University of Technology, February 1987.
- [2] *Controlling the separation of liquids and solids*, Alfa Laval Corporate AB, Rudeboks väg 1, SE-226 55 Lund, Sweden.
- [3] *THE FLOTTWEG DISK STACK CENTRIFUGE*, Flottweg SE, Industriestraße 6-8, 84137 Vilsbiburg, Deutschland (Germany).
- [4] F. Boysa, W. Ayers, and J. Swithenbank, “Fundamental mathematical modelling approach to cyclonyclone design.” *Chem. Eng. Res. Des.*, vol. 60, pp. 222–230, July 1982.
- [5] C. Haig, A. Hursthouse, D. Sykes, and S. Mcilwain, “The rapid development of small scale cyclones — numerical modelling versus empirical models,” *Applied Mathematical Modelling*, vol. 40, pp. 6082–6104, February 2016.
- [6] C. Cortes and A. Gil, “Modeling the gas and particle flow inside cyclone separators,” *Progress in energy and combustion Science*, vol. 33, no. 5, pp. 409–452, 2007.
- [7] A. Nowakowski, J. Cullivan, R. Williams, and T. Dyakowski, “Application of cfd to modelling of the flow in hydrocyclones. is this a realizable option or still a research challenge?” *Minerals Engineering*, vol. 17, no. 5, pp. 661–669, 2004.
- [8] E. Dick, J. Vierendeels, A. Serbrugyns, and J. V. Voorde, “Performance prediction of centrifugal pumps with cfd-tools,” *Task quarterly*, vol. 5, pp. 579–594, 2001.
- [9] K. E. Wardle, T. R. Allen, and R. Swaney, “Computational fluid dynamics (cfd) study of the flow in an annular centrifugal contactor,” *Separation science and technology*, vol. 41, no. 10, pp. 2225–2244, 2006.

- [10] K. E. Wardle, “Open-source cfd simulations of liquid–liquid flow in the annular centrifugal contactor,” *Separation Science and Technology*, vol. 46, no. 15, pp. 2409–2417, 2011.
- [11] B. Yu, “Cfd modeling of two-stage parallel plate sedimentation centrifuge for microalgae dewatering,” Master’s thesis, Department of Sustainable Process and Energy Faculty of Mechanical, Maritime and Materials Engineering in Delft University of Technology., March 2012.
- [12] X. R. Fernández and H. Nirschl, “Cfd simulation of a solid bowl centrifuge used for waste managment,” in *A Numerical And Experimental Study*, 2000.
- [13] M. V. C. Alves, J. R. Barbosa, and A. T. Prata, “Analytical and cfd modeling of the fluid flow in an eccentric-tube centrifugal oil pump for hermetic compressors,” *International Journal of Refrigeration*, vol. 36, no. 7, pp. 1905–1915, 2013.
- [14] L. Dong, S. Fu, and H. Yuan, “Numerical simulation on pressure field in a decanter centrifuge,” *Chem. Ind. Eng. Pro*, vol. 33, no. 2, pp. 309–313, 2014.
- [15] J. Richardson, J. Harker, and J. Backhurst, *Coulson and Richardson’s CHEMICAL ENGINEERING*, 5th ed. Linacre House, Jordan Hill, Oxford OX2 8DP 225 Wildwood Avenue, Woburn, MA 01801-20: Bath Press, Bath, 2002, vol. 2.
- [16] *ANSYS Fluent Theory Guide*, ANSYS, Inc., 275 Technology Drive Canonsburg, PA 15317, November 2013.
- [17] W. R. Gambill, “How to estimate mixtures viscosities,” *Chemical Engineering*, vol. 66, pp. 151–152, 1959.
- [18] H. Versteeg and W. Malalasekara, *An Introduction to Computational Fluid Dynamics, THE FINITE VOLUME METHOD*, 2nd ed. Edinburgh Gate, Harlow, Essex CM20 2JE, England: Pearson Education Limited, 2007.
- [19] *FLUENT User’s Guide*, November 2001, chapter 10 : Modeling Turbulence.
- [20] “Solver settings,” Introductory FLUENT Training, December 2006.

- [21] “Introduction to cfd methodology,introductionto introduction to ansys fluent,” ANSYS Customer Training Material, 2010.
- [22] M. Ozen, “Meshing workshop,” OZEN ENGINEERING, INC, Tech. Rep., November 2014.
- [23] *ANSYS Meshing User’s Guide*, ANSYS, Inc., 275 Technology Drive Canonsburg, PA 15317, November 2013.
- [24] “The density of some common liquids,” November 2015.
- [25] F. Yeaple, *Fluid Power Design Handbook*, 3rd ed. CRC Press, October 1995.
- [26] M. Keating, “Accelerating cfd solutions, several recent enhancements in ansys fluent solver capabilities accelerate convergence and reduce solution time,” in *ANSYS Advantage*, 1st ed., 2011, vol. Volume V, pp. 49–50.
- [27] D. Sofialidis, “Using moving reference frames and sliding meshes,” Introductory Training in ANSYS Fluent, University of Ljubljana, Faculty of Mechanical Engineering, Ljubljana, Slovenia, September 2013.
- [28] *The FLUENT User’s Guide*, Fluent Inc., November 2001, volume 2,Chapter 9,Modeling Flows in Moving Zones.
- [29] F. Russo and N. T. Basse, “Scaling of turbulence intensity for low-speed flow in smooth pipes,” *Flow Measurement and Instrumentation*, vol. 52, pp. 101–114, 2016.
- [30] J. Jaramillo, C. Pérez-Segarra, I. Rodriguez, and A. Olivia, “Numerical study of plane and round impinging jets using rans models,” *Numerical Heat Transfer; Part B: Fundamentals*, vol. 54, pp. 213–237, 2008.
- [31] J. C. Crittenden, R. R. Trussell, D. W. Hand, K. J. Howe, and G. Tchobanoglous, *MWH’s Water Treatment: Principles and Design, Third Edition*,, 3rd ed. Hoboken, New Jersey: John Wiley and Sons, March 2012.

- [32] E. Oberg, F. Jones, H. Horton, H. Ryffel, and C. McCauley, *Machinery's Handbook*, 30th ed., March 2016.