

DAFTAR PUSTAKA

- [1] Y. Daryanto, *Kajian Potensi angin Untuk Pembangkit Listrik Tenaga Bayu*. Yogyakarta: BALAI PPTAGG - UPT-LAGG, 2007.
- [2] R. Indonesia, “Peraturan Presiden Republik Indonesia No. 22 Tahun 2017 tentang Rencana Umum Energi Nasional.” Sekretaris Kabinet Republik Indonesia, Jakarta, 2017.
- [3] S. Nugraha, “Outlook Energi Indonesia 2016.” Sekretaris Jendral Dewan Energi Nasional, Jakarta, 2016.
- [4] Y. Ohya, T. Karasudani, A. Sakurai, K. ichi Abe, and M. Inoue, “Development of a shrouded wind turbine with a flanged diffuser,” *J. Wind Eng. Ind. Aerodyn.*, vol. 96, no. 5, pp. 524–539, 2008.
- [5] T. S. Kannan, S. A. Mutasher, and Y. H. K. Lau, “DESIGN AND FLOW VELOCITY SIMULATION OF DIFFUSER AUGMENTED WIND TURBINE USING CFD,” vol. 8, no. 4, pp. 372–384, 2013.
- [6] F. Bet and H. Grassmann, “Upgrading conventional wind turbines,” *Renew. Energy*, vol. 28, no. 1, pp. 71–78, 2003.
- [7] K. I. Abe and Y. Ohya, “An investigation of flow fields around flanged diffusers using CFD,” *J. Wind Eng. Ind. Aerodyn.*, vol. 92, no. 3–4, pp. 315–330, 2004.
- [8] B. Kosasih and A. Tondelli, “Experimental study of shrouded micro-wind turbine,” *Procedia Eng.*, vol. 49, pp. 92–98, 2012.
- [9] F. Q. Putra, “Analisis Numerik Desain Geometri Diffuser Turbin dengan Penambahan Selubung Inlet dan Tepian,” Universitas Gadjah Mada, 2016.
- [10] M. Nasir, “Analisis Pengaruh Variasi Geometri Diffuser Augmented Wind Turbine dengan Penambahan Lengkungan Inlet dan Tepian (Flange),” Universitas Gadjah Mada, 2017.
- [11] I. B. Dwianditya, “Optimasi Geometri Diffuser Turbin Angin dengan

- Penambahan Lengkungan Inlet dan Tepian (Flange),” Universitas Gadjah Mada, 2017.
- [12] T. H. W. Adhana, N. Gunawan, and A. W. Ruri, “Studi Numerik Pengaruh Geometri dan Desain Diffuser untuk Peningkatan Kinerja DAWT (Diffuser Augmented Wind Turbine),” *J. Tek. Pomits*, vol. 7, no. 7, pp. 1–6, 2014.
- [13] A. M. El-Zahaby, A. E. Kabeel, S. S. Elsayed, and M. F. Obiaa, “CFD analysis of flow fields for shrouded wind turbine’s diffuser model with different flange angles,” *Alexandria Eng. J.*, vol. 56, no. 1, pp. 171–179, 2017.
- [14] L. D. Shinomiya *et al.*, “Numerical Study of Flow Around Diffusers With Different Geometry Using Cfd Applied To Hydrokinetics Turbines,” *22nd Int. Congr. Mech. Eng.*, no. Cobem, pp. 1033–1040, 2013.
- [15] M. Shahsavari, E. L. Bibeau, and V. Chatoorgoon, “Effect of shroud on the performance of horizontal axis hydrokinetic turbines,” *Ocean Eng.*, vol. 96, pp. 215–225, 2015.
- [16] W. Han, P. Yan, W. Han, and Y. He, “Design of wind turbines with shroud and lobed ejectors for efficient utilization of low-grade wind energy,” *Energy*, vol. 89, pp. 687–701, 2015.
- [17] J. Liu, M. Song, K. Chen, B. Wu, and X. Zhang, “An optimization methodology for wind lens profile using Computational Fluid Dynamics simulation,” *Energy*, vol. 109, pp. 602–611, 2016.
- [18] D. Rifai, “Analisis Numerik Diffuser Bertingkat Turbin Angin dengan Penambahan Lengkungan Inlet,” Universitas Gadjah Mada, 2016.
- [19] F. Sorribes-Palmer, A. Sanz-Andres, L. Ayuso, R. Sant, and S. Franchini, “Mixed CFD-1D wind turbine diffuser design optimization,” *Renew. Energy*, vol. 105, pp. 386–399, 2017.
- [20] T. Burton, D. Sharpe, N. Jenkins, and E. Bossanyi, *Wind Energy Handbook*, 2nd ed. Chichester: John Wiley & Sons, Ltd, 2001.

- [21] Y. A. Cengel and J. M. Cimbala, *FLUIDS MECHANICS: FUNDAMENTALS AND APPLICATIONS*, 1st ed. New York: McGraw-hill, 2006.
- [22] J. R. Welty, *Dasar-Dasar Fenomena Transport, Volume I: Transfer Momentum (Terjemahan)*, 4th ed., vol. 1. Jakarta: Erlangga, 2004.
- [23] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics The Finite Volume Method*. London: Longman Scientific & Technical, 1995.
- [24] M. Schäfer, *Computational Engineering – Introduction to Numerical Methods*. Berlin: Springer, 2006.
- [25] C. G. Moreno, *Turbulence, Vibrations, Noise and Fluid Instabilities. Practical Approach*. INTECH, 2010.
- [26] A. Bakker, “Turbulence Models Applied Computational Fluid Dynamics,” pp. 1–47, 2006.
- [27] “Introduction to ANSYS FLUENT.” ANSYS Inc., 2010.
- [28] NAFEMS, “What Is Convergence?,” 2018. [Online]. Available: <https://www.nafems.org/join/resources/cfdconvergence/Page0/>. [Accessed: 01-Apr-2018].
- [29] M. Kuron, “3 Criteria for Assessing CFD Convergence,” 2015. [Online]. Available: <https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/9296/3-Criteria-for-Assessing-CFD-Convergence.aspx>. [Accessed: 01-Apr-2018].
- [30] L. Australia, “Tips & Tricks : Convergence and Mesh Independence Study,” 2012. [Online]. Available: <https://www.computationalfluidynamics.com.au/convergence-and-mesh-independent-study/>. [Accessed: 01-Apr-2018].