

Daftar Pustaka

- Afolabi, Eytayo. A. & Lee, J. G.M. 2013. *CFD Simulation of a Single Phase Flow in a Pipe Separator Using Reynolds Stress Method*. ARPN Journal of Engineering and Applied Sciences Vol 8, No 7, July 2013.
- Akhtari, M., Haghshenasfard, M. & Talaie, M. 2013. *Numerical and Experimental Investigation of Heat Transfer of α -Al₂O₃ / Water Nanofluid in Double Pipe and Shell and Tube Heat Exchanger*. Taylor & Francis Group, LLC.
- Ansys Fluent User's Guide. 2013. Ansys, Inc. USA.
- Behera, Siddharta Shankar. 2013. *CFD Analysis of Heat Transfer in a Helical Coil Heat Exchanger Using Fluent*. Departement of Mechanical Engineering, National Institute of Technology Rourkela.
- Bhanuchandrarao, B. 2013. *CFD Analysis And Performance of Parallel And Counter Flow In Concentric Tube Heat Exchangers*. International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 11, November 2013.
- Ghiaasiaan, S.M. 2008. *Two Phase Flow, Boiling, and Condensation in Conventional and Miniatur System*. Cambridge University Press, Cambridge.
- Mazumder, Quamrul. H. 2012. *CFD Analysis of Single and Multiphase Flow Characteristic in Elbow*. Journal of Scientific Research, Engineering, 2012, 4, 210-214.
- Munson, Bruce. R., Young, Donald. F. & Okiishi, Ted. H. 2013. *Fundamental of Fluid Mechanics 7th Edition*. John wiley & Son, Inc.
- Poursaeidi, Esmaeil. 2011. *Using CFD to Study Combustion and Steam Flow Distribution Effects on Reheater Tubes Operation*. Journal of Fluids Engineering Vol 133.
- Rahul H, Kanade. 2015. *Heat Transfer Enhancement in a Double Pipe Heat Exchanger Using CFD*. International Research Journal of Engineering and Technology (IRJET) Vol 02.
- Song, Shengwei. 2014. *Analysis of Y Type Branch Pipe Exhaust Ventilation Flow Characteristics*. Applied Mechanics ang Materials Vols. 556-562, pp 1054-1058. Trans Tech Publications, Switzerland.
- Sukamta. 2011. *Temperature Profiles Based On Multilocations Of Condensation Of Steam Flow Cooled With Parallel Flowing Water In The Outside Of A Horizontal Pipe*.

Tuakia, Firman. 2008. *Dasar-Dasar CFD Menggunakan Fluent*. Informatika. Bandung.

Versteeg, H. K. 1995. *An Introduction to Computational Fluid Dynamics The Finite Volume Methode*. Longman Scientific & Technical. England