

DAFTAR PUSTAKA

- ANSYS *Fluent Tutorial Guide*, ANSYS, Inc., 2600 ANSYS Drive Canonsburg, PA 15317, 2017.
- Cengel, Yunus., Cimbala J. *Fluid Mechanic Fundamental and Application*, Mc Graw Hill, 2006.
- Granger, B. dan K. Richards. 2016. *Front Wing Design Documentation*, <http://www.eng.nus.edu.sg>, online accessed on 1 April 2019.
- Katz, J. *Race Car Aerodynamic Designing for Speed*, Bentley Publisher, Cambridge, 1995.
- Kusnadi, Muhammad Luthfy. *Simulasi Pengaruh Angle of Attack terhadap Lift Coefficient pada Multi-Element Airfoil Eppler E423 dengan ANSYS Fluent*, Universitas Gadjah Mada, Indonesia, 2016.
- Mouffouk, M. A. 2014. *Aerodynamic Development of The Seguracing F1-R01 Prototype Using CFD*. Thesis. Algeria: University H.L of Batna.
- Raheem, A. T., A. Sapit, dan A. N. Mohammed. 2019. Aerodynamics of Formula One Car Front Wing Cascade Wing during Cornering. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 53 (1) 53 – 60.
- Reddy, J. J. dan M. Gupta. 2006. Finding The Optimum Angle of Attack for The Front Wing of an F1 Car Using CFD. *Proceedings of the 4th WSEAS International Conference on Fluid Mechanics and Aerodynamics, Elounda, Greece, August 21-23: 29-34*.
- SAE, 2019 *Formula SAE Rules, Japan Comp Edition*, Society of Automotive Engineers, Japan 2019.