078-AFD

Computational Investigation of the Aerodynamics of "Formula One" Car Model

Tariq Mostafa Mohammed Al-Sawy

Student, 2nd Year Mechanical Dept., Faculty of Engineering, Zagazig University, Egypt, tariqalsawy7@gmail.com

Supervisor: Prof. Dr. Ahmed Farouk AbdelGawad

Professor of Computational Fluid Mechanics, Vice Dean for Graduate Studies and Research, Zagazig University, Egypt, afaroukgb@gmail.com

The most important consideration in "Formula One" (F1) car design is Aerodynamics. "Aerodynamics" is the way air moves around things [1]. It is the difference between championship challenging car and a tail car [2]. Simply, F1 car aerodynamics design has to consider primary concerns: minimizing drag resulting from air resistance; and increasing downforce to push the car tires onto the track and stabilize the car during cornering. This paper discusses the aerodynamics and the resulting forces on F1 car body using CFD. Solidworks 2015 software [3] was employed to create the 3D model used in the simulation. The model was available on GrabCAD [4]. Also, ANSYS Fluent, v.17.1 [5] was used to simulate and analyze the aerodynamics of the car when it is running in a straight line, and cornering. Drag and lift coefficients, velocity streamlines and pressure contours were calculated and presented as results of this simulation.