

Numerical Study on Parallel Rear-Spoiler on Super Cars

Authors : Anshul Ashu

Abstract : Computers are applied to the vehicle aerodynamics in two ways. One of two is Computational Fluid Dynamics (CFD) and other is Computer Aided Flow Visualization (CAFV). Out of two CFD is chosen because it shows the result with computer graphics. The simulation of flow field around the vehicle is one of the important CFD applications. The flow field can be solved numerically using panel methods, k- ϵ method, and direct simulation methods. The spoiler is the tool in vehicle aerodynamics used to minimize unfavorable aerodynamic effects around the vehicle and the parallel spoiler is set of two spoilers which are designed in such a manner that it could effectively reduce the drag. In this study, the standard k- ϵ model of the simplified version of Bugatti Veyron, Audi R8 and Porsche 911 are used to simulate the external flow field. Flow simulation is done for variable Reynolds number. The flow simulation consists of three different levels, first over the model without a rear spoiler, second for over model with single rear spoiler, and third over the model with parallel rear-spoiler. The second and third level has following parameter: the shape of the spoiler, the angle of attack and attachment position. A thorough analysis of simulations results has been found. And a new parallel spoiler is designed. It shows a little improvement in vehicle aerodynamics with a decrease in vehicle aerodynamic drag and lift. Hence, it leads to good fuel economy and traction force of the model.

Keywords : drag, lift, flow simulation, spoiler

Conference Title : ICAH 2015 : International Conference on Aerodynamics and Hydrodynamics

Conference Location : London, United Kingdom

Conference Dates : September 25-26, 2015