

CFD Investigation of the effect of crank geometry and pedalling dynamics on drag.

Abstract

Computational Fluid Dynamics (CFD) is used to investigate the aerodynamic characteristics of a range of bicycle crank designs in clean form and including rider lower leg and foot. The approach makes use of a non-peddalling approach and targets crank and foot positions throughout the pedalling stroke at intervals of 22.5 degrees and is based upon a RANS approach. Flow fields are used to evaluate the wake structures and drag production due to interaction of crank and foot. Force data identifies that drag associated with clean cranks is minimum when the cranks are aligned horizontally with the majority of drag production being associated with the structures located at the bottom bracket centre throughout the pedalling stroke. Wake plots highlight that the design of the crank spider and mounting of the chainrings plays a key part in the production of resistance, reduction in throughflow across the crank spider is observed to be linked with reduced drag. It is observed that solid chainrings offer 36% reduction in average drag coefficient over traditional road style double cranksets in a clean form. Modelling the interaction of the crank with athlete foot, using a standard road crank, highlighted a shift in the minimum drag position, moving to approximately 210 degrees of driveside rotation compared with 180 degrees for the clean crankset. The findings highlight possible areas to optimise crank position when freewheeling. Further work focuses upon the effect of interaction of the foot and motion of the crank during rotation to capture the effects of relative flow speed differences during rotation.

Keywords

CFD, Aerodynamics, Drag, Crank