CFD Analysis of PACE Formula-1 Car

Computer-Aided Design and Applications
Pages:
1-14
Issue:
1
Date:
Thursday, August 29, 2013

An analytical study on air flow effects and resulting dynamics on the PACE Formula 1 race car is presented. The study incorporates Computational Fluid Dynamic analysis and simulation to maximize down force and minimize drag during high speed maneuvers of the race car. Using Star CCM+ software and mentoring provided by CD ? Adapco, the simulation employs efficient meshing techniques and realistic loading conditions to understand down force on front and rear wing portions of the car as well as drag created by all exterior surfaces. Wing and external surface loading under high velocity runs of the car are illustrated. Optimization of wing orientations (direct angle of attack) and geometry modifications on outer surfaces of the car are performed to enhance down force and lessen drag for maximum stability and control during operation. The use of Surface Wrapper saved months of time in preparing the CAD model. The transform tool and Contact Prevention tool in STAR-CCM+ proved to be an efficient means of correcting and modifying geometry instead of going back to the CAD model. The CFD simulations point out that the current front and rear wings do not generate the desired downforce and that the rear wing should be redesigned.

Author Name:
Satyan Chandra, Allison Lee, Steven Gorrell and C. Greg Jensen
Author Company:
Brigham Young University
CAD_PACE_1__1-14.pdf[1]
Industries:
Sports[2]
Products:
STAR-CCM+®[3]

Links:
[1] http://www.cd-adapco.com/sites/default/files/journal/pdf/CAD_PACE_1__1-14.pdf
[2] http://www.cd-adapco.com/industries/sports

[3] http://www.cd-adapco.com/products/star-ccm%C2%AE