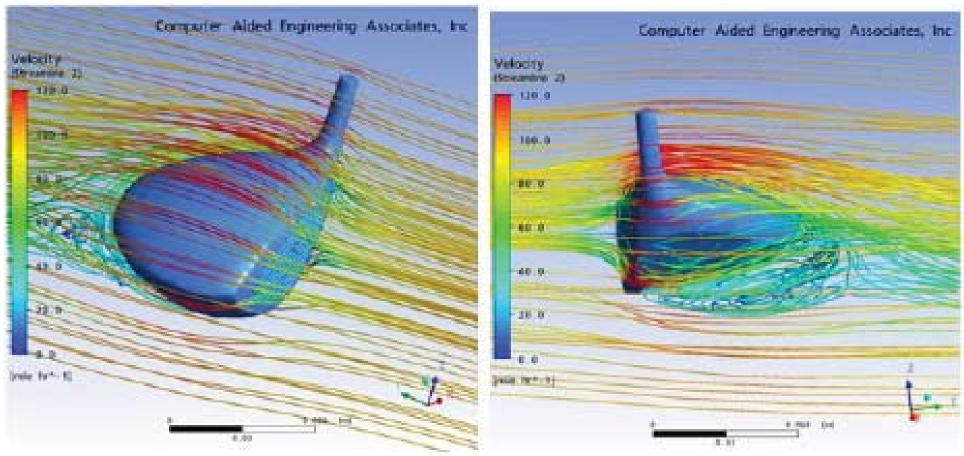
CAE ASSOCIATES
ENGINEERING ADVANTAGE

Case Study



Representative streamlines and flow-field solutions for a driver.

Golf Club Drag Force CFD Analysis

Given today's development trends of maximizing the golf club volume to increase the size of "sweet spot" on the driver face, golf club manufacturers are often interested in predicting drag forces and identifying specific geometric features contributing to the total drag (pressure and friction drag) on drivers.

With club head speeds exceeding 100 mph prior to contact with the golf ball, a driver is a bulky, intrusive object which can generate significant drag force during the swing motion. The shape of the club has a definite influence on this force, and golf club designers must account for this while trying to optimize the club shape.

One method for computing golf club drag is to build a variety of prototypes and conduct wind tunnel testing. At upwards of $1,500 per hour to rent a wind tunnel, with many different orientations and speeds to consider, prototype testing becomes a very expensive and time-consuming option. Also, very little useful information is provided as to why one design is better than another.

Computational Fluid Dynamics (CFD) analysis and consulting is an attractive alternative to wind tunnel testing for this application. Not only can many different configurations and orientations be easily analyzed, but the features creating the most drag can be readily identified from the results.

www.CAEAI.com

Continues >

1579 Straits Turnpike, Suite 2B / Middlebury, CT 06762 / P 203.758.2914 / F 203.758.2965

Golf Club Drag Force CFD Analysis / *Continued*

With these benefits in mind, our client asked CAE Associates' CFD consultants to perform a CFD analysis by conducting an aerodynamic performance simulation for two candidate driver designs. The primary goal of the CFD analysis was to compare the drag force between the two configurations with a nominal speed of 100 mph (just prior to contact with the golf ball).

The CFD analysis began by utilizing the ANSYS/CFX suite of CFD software for this application, which supports parametric associativity with the driver geometry file. This allows for easy geometric changes and "what-if" studies critical during the early phases of the design process.

Based on the shape of the club, it was anticipated that there would be a wake region behind the club, which includes recirculation, shear layer mixing and possibly vortex shedding. For this type of analysis, CFD turbulence modeling is always a concern. The standard k-epsilon model is not ideal for separated flows, as the results of the aerodynamic performance simulation tend to overpredict the drag. Instead, turbulence modeling via the SST (shear stress transport) method was used in order to capture the adverse pressure gradient in the wake region better. The SST model has demonstrated the capability to predict flow separation and shear layers better than all other two-equation models used in turbulence modeling.

The CFD analysis provided the resultant drag force for both designs. More importantly, it displayed the three-dimensional flowfield solutions to help identify the contributing factors to the differences in drag between the two candidate designs. The client was able to use the information derived from the CFD analysis to decide which design was best to move forward with, and which modifications would reduce the drag force even further.

For more about CAE Associates' CFD consulting services or to learn about turbulence modeling from our team of CFD consultants, please contact our office at 203-758-2914.

1579 Straits Turnpike, Suite 2B / Middlebury, CT 06762 / P 203.758.2914 / F 203.758.2965