PROFESSIONAL MOTORSPORT WORLD

pmw

APRIL-JUNE 2012

Magic formula
Can Formula 3 withstand the challenge from GP3?

Get your coat
The latest developments in high-performance coatings

Perfect match?
Toyota builds a petrol-electric hybrid for Le Mans

DeltaWing

Half the weight and half the power of a regular Le Mans Prototype. Is this the future of sportscar racing?

Interviews

Roger Griffiths
Technical director, HPD

Paul Hembery
Motorsport director, Pirelli

Beth Paretta
Marketing and operations director, SRT

THE MAGAZINE FOR THOSE IN THE BUSINESS OF GOING MOTOR RACING

WWW.PMW-MAGAZINE.COM

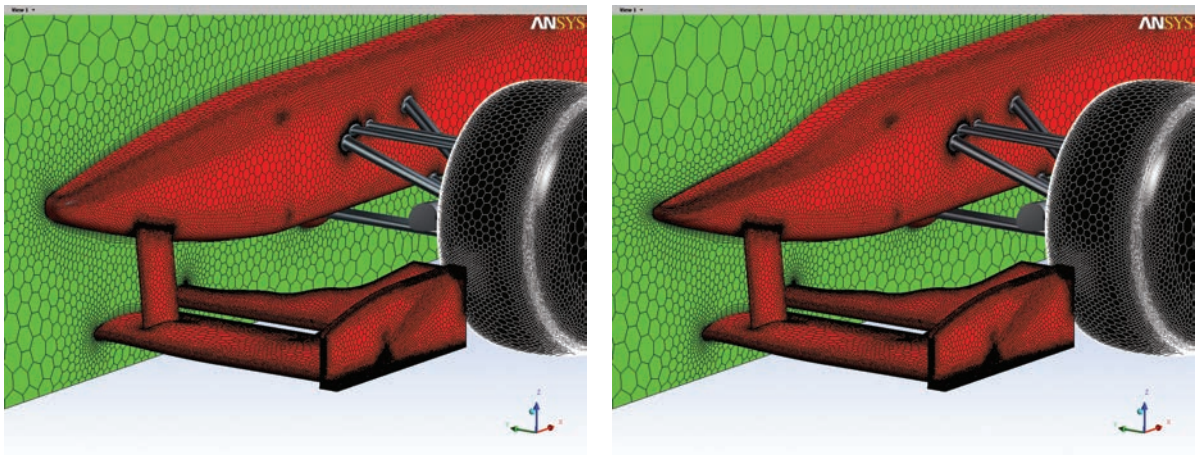

Figure 2: ***Before and after morphing into 'Ugly Nose' configuration with RBF Morph***

HIGHER-SPEED CFD

The latest engineering simulation software from **ANSYS** is making it easier for motorsport aerodynamicists to trial more options, faster, in the pursuit of optimal performance

The recent launch of ANSYS 14.0 engineering simulation software brought exciting new functionality and gains in high-performance computing.

Prior to this release, ANSYS tools already achieved excellent linear scalability on computer clusters to around 2,000 compute cores; that is, 2,000 cores could complete a CFD calculation in 1/2,000th the time required on a single core. As the number of cores increased from that point, efficiency was seen to diminish. ANSYS 14.0 shows linear scalability up to 3,000 cores on a number of test cases, including an open-wheeled race car containing 130 million computational cells (Figure 1).

As CFD case sizes have increased from tens to hundreds of millions of cells, simulation file sizes have increased to tens of gigabytes of data per simulation. Furthermore, demanding F1 teams have identified the process's bottleneck to be the time needed for software to read and write these files (file i/o). ANSYS developers have worked hard to address this issue, with the time taken to write data on a massive 750-million cell model being reduced from over an hour in previous versions, to only five minutes using 'parallel file i/o' in more recent versions of the code.

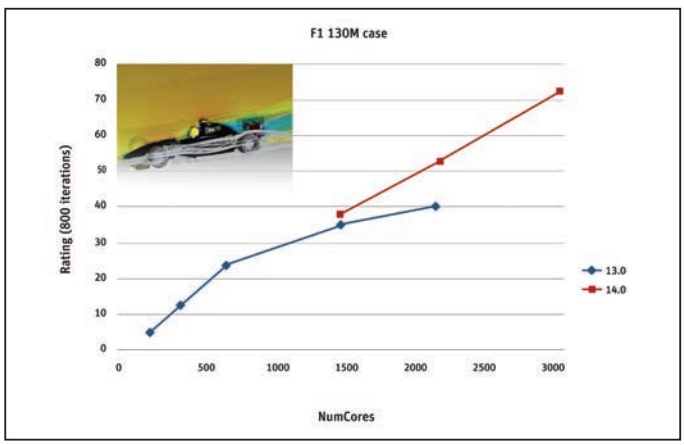

Figure 1: ***Linear scalability on a large open-wheel race car CFD calculation in ANSYS FLUENT 14***

"RBF MORPH IS AN INGENIOUS MORPHING TOOL THAT ALLOWS ENGINEERS TO MOLD THE GEOMETRY LIKE CLAY TO VERY HIGH PRECISION"

With such high CFD solver speeds, the onus falls on the aerodynamicist to 'feed the furnace' with as many car designs as possible as he strives to find optimal performance. The bottleneck here is the design and creating the computational grid, which can take many hours. ANSYS software has now addressed this issue by placing a unique morphing tool – called RBF Morph – into the hands of CFD engineers. It is the brainchild of Marco Biancolini of the University of Rome.

RBF Morph is an ingenious morphing tool that sits inside the CFD solver and allows engineers to mold the geometry like clay to very high precision. The technology can be used during parallel computations on high-performance computing clusters to parametrically alter the geometry (for example, change wing angles, wheel steer, car ride heights) quickly and efficiently with no need for regridding the geometry or expensive file i/o.

The only problem at this point is that the realm of possibilities becomes too great for the CFD aerodynamicist! How does he know which combination of design tweaks will lead to an overall increased efficiency of the aerodynamic package?

ANSYS has the answer with its automatic optimization tool – ANSYS DesignXplorer. Given a set of input parameters, such as wing angle of attack, DesignXplorer automatically creates a design of experiments to efficiently explore the whole design space and then find the optimal design from the simulation outputs using a range of statistical tools.

At ANSYS 14.0, the coupling of RBF Morph with DesignXplorer opens up the possibility to do fully parametric, automatic optimization of fully detailed F1 cars with the highest fidelity CFD code on the market. <

Contact

Andy Wade, senior technical services engineer, ANSYS UK
Tel: +44 114 281 8888;
Email: andy.wade@ansys.com;
Web: www.ansys.com

003 READER ENQUIRY REFERENCE