Formula SAE competition challenges university students each year to design, build, market and race a small, open-wheeled, Formula-style car against other such institutions from around the globe. Monash Motorsport, from Monash University in Melbourne, Australia, is one of these teams.

The Monash Motorsport team comprises approximately 70 undergraduate students, primarily from the Department of Mechanical and Aerospace Engineering but also from disciplines such as science, business, marketing and even law. The demanding nature of the competition gives students the chance to develop important skills in teamwork, communication and project management, along with helping them to prepare for the challenges they will face when they embark on professional careers in industry.

Engineering students who participate in this program benefit greatly from the opportunity to develop their expertise in computer-aided design and engineering (CAD and CAE) by modeling and simulating many different components and systems within the Formula SAE car. Monash Motorsport team members have utilized ANSYS engineering simulation software for more than a decade to accomplish this work.

Through a close relationship with the local ANSYS channel partner, LEAP Australia, the group developed a range of tutorials to help team members and other students conduct finite element analysis (FEA) and computational fluid dynamics (CFD) studies for common Formula SAE applications. Each year over spring break, the Monash team organizes a three-day symposium called Design to Win, during which local Formula SAE teams receive training on ANSYS software and present examples of their work.

Monash Motorsport recently finalized design and development of its latest racer, the M13. The team's cars are well known for their distinctive aerodynamic packages; Monash claimed four competition wins and several top-five places in recent Australian, U.K. and German events. Aerodynamic packages in Formula SAE are becoming popular, as teams learn that wings do indeed offer...
ACADEMIC STUDENT TEAM

The Monash Motorsport team won its fifth consecutive Australasian FSAE Championship in 2013, with a strong performance on track and in all static events. The team will also compete with their 2013 car at Formula Student UK and Germany in 2014, hoping to improve upon third- and fourth-place finishes in these events in 2012.

AERODYNAMIC IMPROVEMENTS

The new M13 racer is a clean-sheet redesign in all respects, incorporating a number of significant aerodynamic improvements and novel design features made possible by extensively using ANSYS Mechanical, ANSYS CFX and ANSYS Fluent. The car is one of only a few Formula SAE vehicles worldwide to utilize a drag-reduction system (DRS), used in current Formula One racing. This innovation enables the angle of the flaps in the multi-element front and rear wings to be dynamically adjusted via pneumatic cylinders and linkages. As a result, the car has two distinct aerodynamic modes: high downforce and low drag.

The DRS is activated when the driver presses a button on the steering wheel, so the low-drag setting can be used when the car is driving in a straight line and significant downforce is not required. By using a button to engage DRS, the driver can revert to the high-downforce mode and maximize the car’s downforce (and drag) before applying the brakes at the end of a straight — which an automated system cannot do reliably without GPS-enabled track mapping.

Having access to a drag-reduction system has allowed this year’s Monash team to significantly increase its downforce target for the M13 car to a CL.A (“A” denotes frontal area — when combined with lift coefficient, this provides a more representative measure of the car’s performance on track while taking geometry into account) of greater than 6, given that drag is no longer a significant limitation on straight-line performance. Before starting design, the team invested time in developing a standard fluid domain and boundary setup to ensure consistency and comparability among all future simulations. Domain size and mesh sensitivity studies were undertaken, and benchmarking tests were conducted with different turbulence models.

The team chose to use a symmetry model (using only half the car) to maximize mesh resolution, given RAM-based meshing and solution time limitations when working on single local nodes. Testing showed that approximately 15 million to 20 million elements for

ANSYS CFX results demonstrate the dramatic differences in surface pressure magnitude and vortex structures (iso-surfaces) generated by high-downforce mode (left) and DRS-engaged low-drag mode (right). DRS activation results in a 50 percent reduction in drag generated by the full car.

© 2014 ANSYS, INC.
the symmetry model provided the best compromise between mesh resolution and solver time, based on Monash’s current computational resources. A 400-iteration run using the k-omega SST turbulence model generally solved in less than six hours, which was considered an acceptable turnaround time.

The eight-person aero team conducted and documented almost 200 unique aerodynamic design iterations over a three-month period at the start of 2013. A team-developed ANSYS CFD-Post state file was used to allow fast and consistent automated post-processing as well as output of figures (pressure contours, streamlines and vortex cores), tables of force and coefficient results via the report function. The team utilized custom pressure color scales to clearly differentiate positive pressures (yellow to red) from suction pressures (blue). A custom red/blue scale was applied to stream-wise vorticity and used to color vortex core iso-surfaces, neatly highlighting the direction of vortex rotation. Chord-wise plots of coefficient of pressure for the front and rear wings at a range of span-wise locations were routinely generated to fine-tune wing profiles as well as to better understand span-wise pressure variations.

Keyframe animations were used extensively to generate longitudinal total pressure sweep videos and vortex core videos, providing insight into the complex vortex and wake interactions that dominate the vehicle’s near field. Juggling these vortex and wake interactions proved crucial in maximizing the downforce produced by the front wing and underbody diffuser, as well as in balancing the front and rear downforce distribution for the whole car. These full-car interactions drove the team’s final choice of rear-wing height and provided confidence with respect to the cooling flows entering the radiator and turbo intercooler.

The full results from each run, along with associated CAD models and ANSYS Workbench archives, were updated to a private team Wiki in real time throughout the design phase, which facilitated rapid communication and results sharing among the team. This ensured that all members remained updated on the design progress, which minimized repetition and duplication as well as helped...
student engineers to discuss and incorporate the best design features into the next round of CFD runs.

**INFRASTRUCTURE FOR COMPLEX SIMULATIONS**

Beyond conducting CFD analysis, the Monash team developed a methodology and hardware infrastructure required to conduct large and complex simulations (incorporating up to 200 million elements) combined with a rotating reference frame. A rotating reference frame is needed for modeling aerodynamic effects when the car is turning a corner, since the interactions cannot be accurately estimated nor understood using traditional fixed-flow yaw angles applied to the entire car (as in a wind tunnel).

The team developed a 100-node local Beowulf-style cluster by utilizing idle desktop machines in the Monash Engineering Computer Labs, which were made available for the team’s use overnight and on weekends. A fully automated grid-generation outsourcing tool was scripted to allow geometry clean-up, surface and volume meshing, and solving to be completed remotely on the cluster, thereby avoiding RAM limitations and slow transfer times for the large meshes, which otherwise would be generated locally.

The incident angle, $\theta$, is the angle that the freestream air makes with the car centerline at the point of impact. The freestream vector is tangential to the center of rotation and, therefore, perpendicular to any line that radiates from the center of rotation. The angle is identical to that formed between the line radiating from the center of rotation to the point of interest and the line that radiates from the center of rotation and is perpendicular to the centerline of the car. By decreasing the parameter $r$, both $\theta_{\text{front}}$ and $\theta_{\text{rear}}$ increase. Increasing the parameter $\psi$ moves the center of rotation point rearward. This has the effect of reducing the rear incidence angle, $\theta_{\text{rear}}$, but increasing the front incidence angle, $\theta_{\text{front}}$. Due to the large cost in time of setting up, solving and post-processing a rotating reference frame simulation on the cluster, only one case was considered. The 200 runs conducted by the team were completed for a straight line case, and automation allowed runs to be turned around in 12 hours. Using this method, the team could cycle through many different iterations within a very narrow design window of approximately three months.
WIND TUNNEL EXPERIMENTS
Monash Motorsport is fortunate to have access to a full-scale automotive wind tunnel on campus; this has allowed the team’s engineers to extensively correlate aerodynamic predictions obtained from ANSYS software with data from on-track testing. Typically, the team starts by correlating wind tunnel and CFD results for performance of wings in isolation, via wing angle-of-attack and yaw angle sweeps made in freestream in the tunnel. Then the car is added, which allows a detailed study of the rear wing height along with endplate size, shape and detail features. Cooling performance is measured using a specific-dissipation test rig within the tunnel, which circulates heated coolant through the radiator at a measured flow rate. Thermocouples in the coolant lines allow the team to calculate heat dissipation as a function of the temperature differential between ambient air flow and coolant.
SUMMARY

CFD has proven to be a powerful tool for the Monash Motorsport team, particularly since wind tunnel testing time is limited to a few days each year. The team can narrow down the most promising design concepts without having to incur the cost of fabricating each design change and physically testing it in the wind tunnel or on track. Furthermore, automation of the simulation setup in ANSYS software has allowed for quicker turnaround times on simulations (down to 12 hours from 24 hours), and standardized report generation has yielded significant improvement in team knowledge transfer (with reports of each run saved on the team knowledge database for future members to access and learn from) as well as ease of comparison between runs. The use of CFD has allowed the team to spend financial and time resources for building and testing various prototype designs on only the most promising few.

Authors’ Note

Monash Motorsport has published several SAE papers on the aerodynamic development of its past cars, and team members are happy to talk with other teams implementing aerodynamic studies. The team sincerely thanks all current and past team members for their hard work and dedication to this project — as well as LEAP Australia, ANSYS, the Department of Mechanical and Aerospace Engineering at Monash, and the Monash wind tunnel facility.