



CFD Testing in F1 In Schools

Engineering Education in Secondary Schools

Singapore

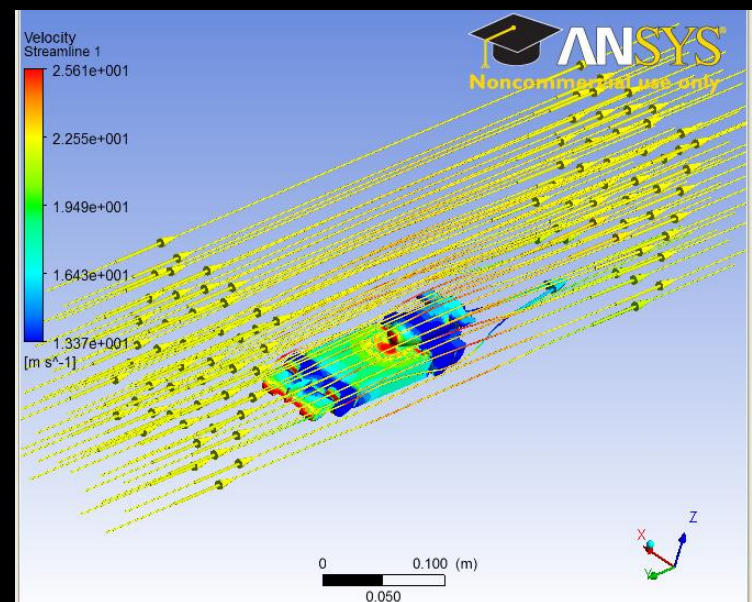
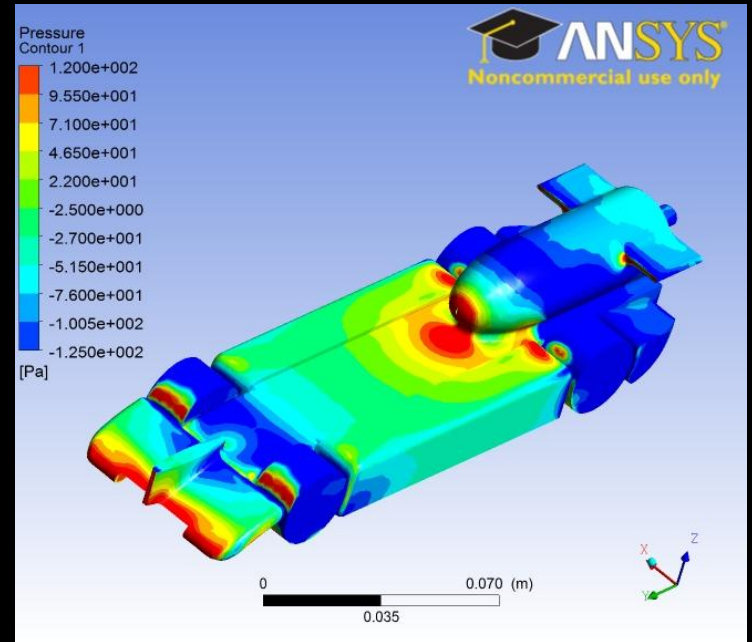


Overview

F1 in Schools is the global multi-disciplinary challenge in which teams of students aged 9 to 19 deploy CAD/CAM software to collaborate, design, analyse, manufacture, test, and then race miniature gas powered balsa wood F1 cars. It is a unique global platform for the promotion of Formula One and partners to a youth market. Teams must raise sponsorship and manage budgets to fund research, travel and accommodation.

The challenge inspires students to use IT to learn about physics, aerodynamics, design, manufacture, branding, graphics, sponsorship, marketing, leadership/teamwork, media skills and financial strategy, and apply them in a practical, imaginative, competitive and exciting way.

In the testing of the aerodynamics of our car, we have chosen ANSYS as our main CFD software in the testing of our draft designs



Testimonial

ANSYS CFD software allowed us to input more realistic data such as rotational speed of the wheels as well as pressure of the gas canister. This allowed our design engineer to have a more realistic simulation to our actual car performance as well as animations for our students to understand the pressure and velocity profiles of the car.

Mr Kenneth Tan Chin Beng
Teacher (F1 in Schools)
Henderson Secondary School

Process

The analysis of the velocity flows around the car is crucial to the speed of the car. It is not possible to manufacture every design to test the finished car and thus, our team rely heavily on ANSYS CFD to analysis the performance of the car (Drag coefficient, Velocity, pressure profile) redefine the design before manufacturing.

Solution

The use of ANSYS CFD allows the team to see which parts of the car is performing well and which parts of the car is not. But changing a few parameters in the software, we were able to test more designs in a shorter time thus increasing our design efficiency.

Benefits

ANSYS allows the students to get the drag coefficient of the various parts of the car (something which the old software was not able to handle). We were able to see a visual animation of the velocity streamlines going around the car to determine which parts of the car is causing turbulence