



Virtual Wind Tunnel

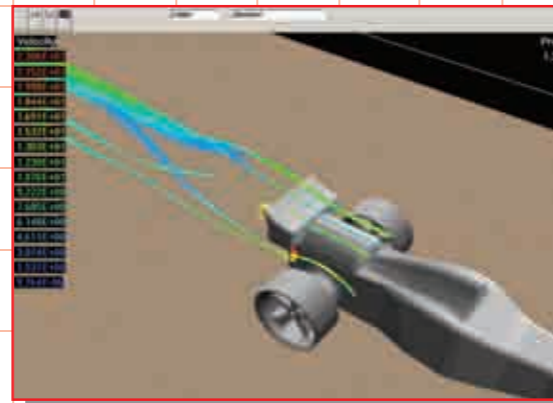
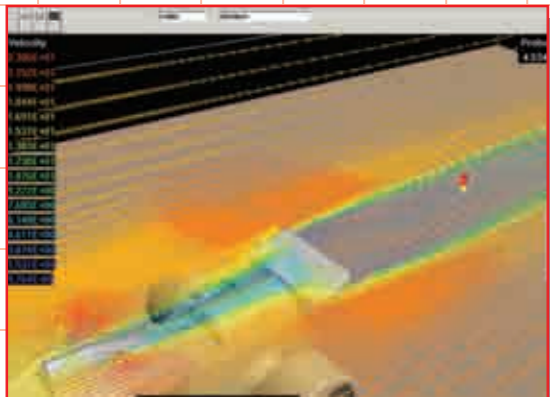
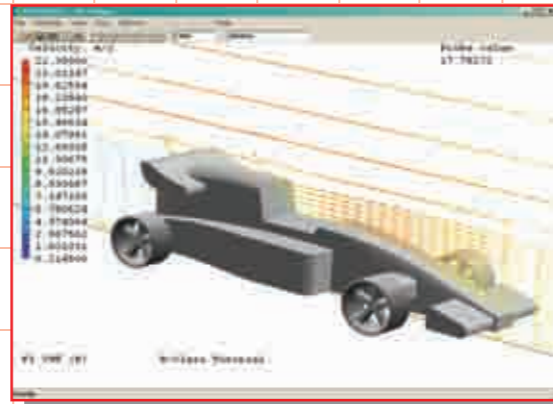
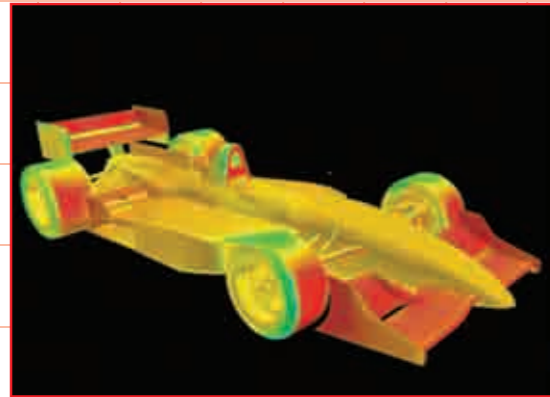
F1 VWT ANALYSIS SOFTWARE



Virtual Wind Tunnel

F1 VWT ANALYSIS SOFTWARE

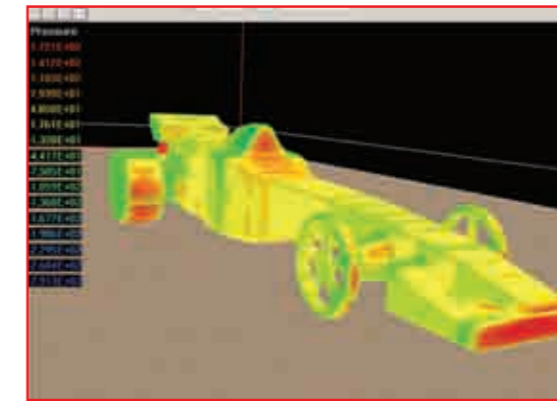
The Virtual Wind Tunnel provides an innovative and cost-effective way for schools and colleges to analyse and test their F1 cars. The results will provide you with information relating to the potential performance of your design, and provide guidance towards the areas for improvement.



For those involved in the F1 Challenge, the process is simple - students design their F1 car with 3D CAD software such as SolidWorks, and then export the STL file into the virtual wind tunnel software. The design is then displayed on-screen, allowing students to begin testing the designs for velocities, pressures, areas of turbulence, lift and drag by using vector plots, contour plots, streamlines and isosurfaces.

The Virtual Wind Tunnel Software uses a process called Computational Fluid Dynamics or CFD. This is basically the prediction of processes involving fluid flow, heat and mass transfer, chemical reaction and/or combustion. Anything that involves fluid flow can be simulated using these techniques, with varying degrees of accuracy.

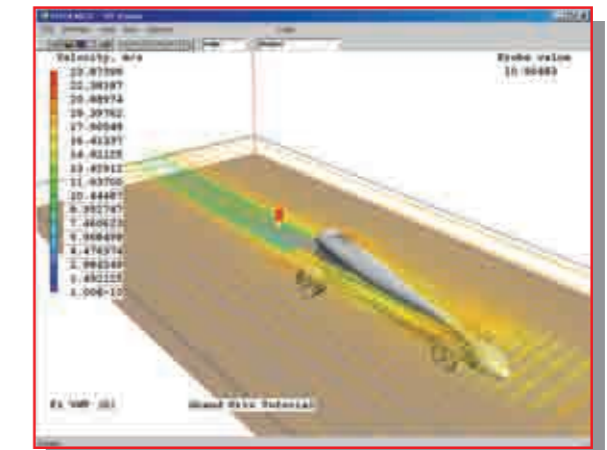
CFD is based upon the laws of physics, of conservation of mass, momentum and energy. The equations are embodied within a mathematical model and solved using a grid superimposed on the region of interest. For the F1 in Schools Challenge, this will be your "analyse process" - i.e. Step 2 of your team's progress towards racing success.



Your design will be imported from CAD software and displayed in our Formula 1 Virtual Wind Tunnel (F1 VWT) software which is already set up to receive it. You will alter initial settings, boundary conditions and other factors in the F1 VWT pre-processor, before starting the mathematical 'solver'. Once the solution has been reached, you will visualise the results interactively in graphical form, using the F1 VWT post-processor.

You will see velocities, pressures, areas of turbulence, lift and drag, using vector plots, contour plots, streamlines and iso-surfaces. The forces on the car body will be calculated and plotted to provide lift and drag data.

Industrial design companies use CFD techniques to assist with their prototype designs. What you do with the information is up to you ...



SYSTEM REQUIREMENTS

Any standard Windows PC (XP or VISTA) system. The software is both CPU- and RAM-intensive but 1GB RAM should suffice (& 2GB is better), and a 1GHz processor speed (as a minimum). No special graphics requirement. Both 32-bit and 64-bit systems are supported but, if required, 64-bit must be specified at time of order.



Get help, advice and share designs online
www.denforddata.com/bb