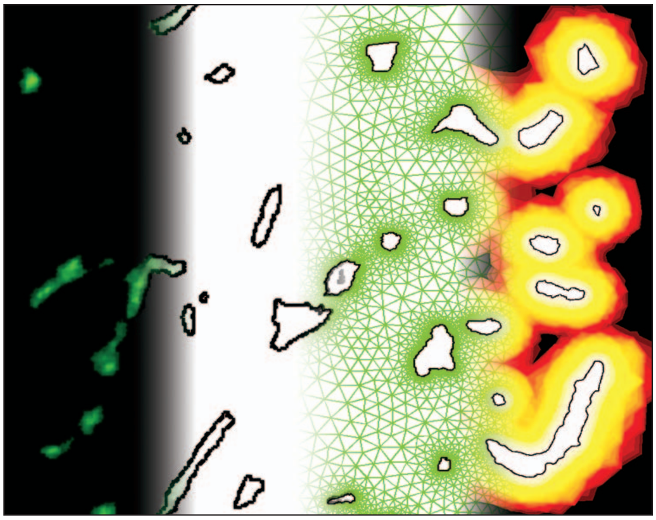
Having Fun While Studying CFD

By André Bakker, Fluent Inc. and Dartmouth College, Hanover, NH

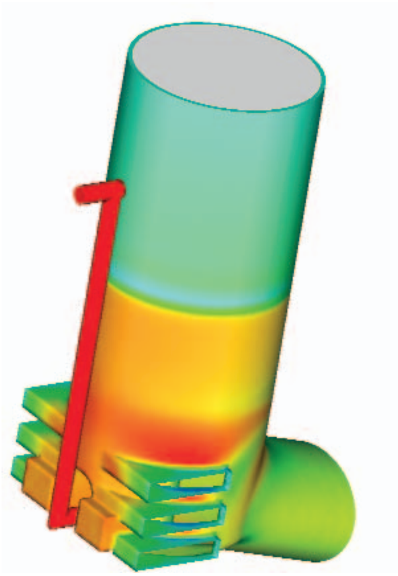
THE THAYER SCHOOL OF ENGINEERING at Dartmouth College has offered a graduate class on CFD for several years now. Students learn about the theory behind fluid flow and heat transfer, turbulence modeling, and numerical methods. The class ends with the students choosing a research topic of their own interest, and studying it using CFD. A wide range of topics is usually covered in the projects, and highlights from some recent projects are summarized below.



A sequence of tumor images shows (from left to right) the fluorescent image, the outline of the blood vessels, the computational mesh, and the drug volume fraction after a period of time has passed

Drug diffusion in a tumor

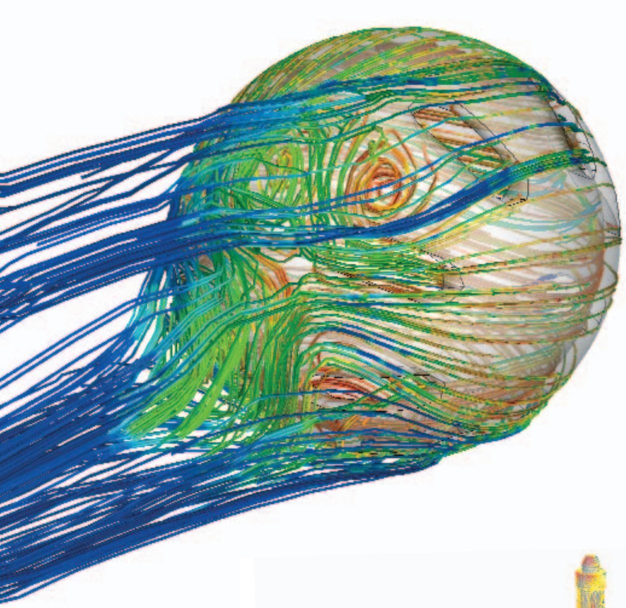
Xiaodong Zhou and **Rajeev Kumar** studied the diffusion of a drug inside a tumor. Their objective was to develop a model to better understand the physics behind this type of drug delivery. In order to obtain a realistic model of a tumor, the students took fluorescent images of a slice of a frozen mouse tumor to determine the locations of the blood vessels. These images were converted to contour images in MATLAB®. Edges and vertex data were extracted, and imported into GAMBIT, where a 2D mesh of the tumor cross-section showing the locations of the blood vessels was created. Transient calculations of the diffusion of the drug from the blood vessels into the solid part of the tumor were then performed using FLUENT. Drugs are active chemicals that are not just absorbed, but consumed because they are bound by the tumorous cells. The ratio between the rates of consumption and diffusion determine how deep a drug will penetrate into a tumor. It is important that the drug reach all regions of the tumor, and this process is controlled by preparing the drug in special ways, such as encapsulating the active drug in very tiny fat particles. This fat-encapsulated drug is absorbed better and its distribution to the tumor site is improved. Xiaodong and Rajeev developed user-defined functions (UDFs) for different consumption models and evaluated their effect on the distribution of the drug in the tumor. They found that realistic results could be obtained using a first-order model for the drug consumption.



Contours of static pressure on the surface of an inhaler equipped with a spirometer; the pressure sensor is shown in red

Pressure in an inhaler with a spirometer

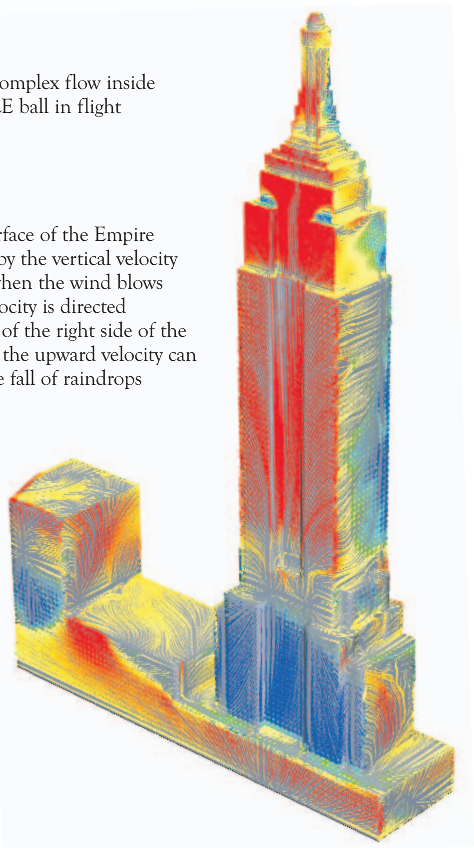
Jordan Desroches analyzed a portable spirometer that is integrated into a standard medication inhaler. Millions of people suffer from asthma and rely on inhalers to dispense their medication. About 25% of them can not tell when they are becoming short of breath, however, a condition called "low perception of dyspnea." Spirometers allow them to assess how well their lungs are working by measuring the maximum volume of air that they can exhale. The user blows into the inhaler equipped with a spirometer; a pressure sensor picks up the resultant pressure pulse and determines the user's pulmonary status from this data. The inhaler has gills to vent excess pressure during exhalation. Jordan was interested in sizing the gills correctly so that the pressure sensor gives an accurate measure of the pressure pulse. This would allow the measured pressure to be correlated with the exhalation velocity. Furthermore, he wanted to prevent recirculation zones or other flow features that might result in inaccuracies in the pressure pulse measurement. Jordan created the geometry in Pro/ENGINEER, meshed it with GAMBIT, and performed transient simulations of the pressure at the sensor as a function of the exhalation velocity of the user with FLUENT. He found that there was a good correlation between the measured pressure and flow rate. This information can now be used to calibrate the electronics in the device.



Pathlines illustrate the complex flow inside and outside of a WIFFLE ball in flight

WIFFLE® ball

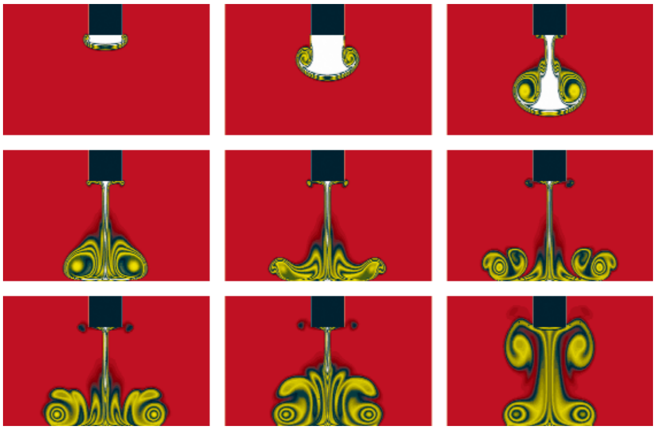
John Gagne and **Alex Tee** were interested in the flow field through and around a WIFFLE ball, a hollow plastic ball with several openings, popular for use in outdoor games. The ball can be thrown with very strong curves when the user gives it the right spin. This is thought to be due to the aerodynamic phenomena resulting from the flow through the spinning holes. John and Alex tried several options to model the flow around the ball. They found that in order to obtain good results they had to use the sliding mesh model to include the transient phenomena resulting from the rotation of the holes. A strong circulation flow was found to exist inside the ball, as well as vortices in the wake. The exact nature of the curved path the ball takes when thrown is most likely due to the interaction between these different vortex structures.



Oil film lines on the surface of the Empire State Building, colored by the vertical velocity component show that when the wind blows from the left, the air velocity is directed upwards along portions of the right side of the building; in some cases, the upward velocity can be enough to reverse the fall of raindrops

Empire State Building

Exchange students from the University of Aachen in Germany **Philip Engelhardt** and **Daniel Wichmann** wanted to model an American icon: the Empire State Building (ESB) in New York. According to popular lore, during rainstorms there are windows in this building where people claim to see raindrops going up instead of down. Philip and Daniel concluded that CFD offered an excellent way to test this theory. They studied maps of New York, aerial photographs, and building drawings to create a GAMBIT model of both the ESB and the closest surrounding buildings. They also studied meteorological data to obtain realistic vertical profiles for the local wind velocity. They studied a variety of wind conditions and raindrop sizes. They concluded that it is indeed possible for the upward velocity of the wind flowing over the building walls and windows to exceed the falling velocity of raindrops. So, yes, it is possible that one day you will find yourself looking out over New York and wondering why the rain seems to be falling up instead of down!



Tracer concentration field in an impinging jet at a Reynolds number of 1600 as a function of time; the image sequence covers a total of 16.2 seconds

Impinging jet

Arthur Shaw and **Robert Haehnel** were interested in studying the flow field of impinging jets, particularly those in the downwash of helicopter blades. For validation purposes, they studied the spreading rate of a round jet, and also reproduced an experimental flow visualization study. In the experiment, the flow field of a pulsed jet at a Reynolds number of 1600 is visualized by means of fluorescent tracers and a laser sheet, showing the vortex rings that form, interact, and eventually break up. In the simulation, a tracer species was used to visualize that same flow field. They performed a detailed comparison between the experimental images and the FLUENT results, and concluded that all key flow field features were resolved. Satisfied with the results of the validation study, they studied the dispersion of dust and sand particles in an impinging jet, as it would occur below a helicopter, which is important because it affects the pilot's visibility during take-off and landing.

As always, the students were very creative and showed that CFD is not just a practical engineering tool but can also be fun to use! ■