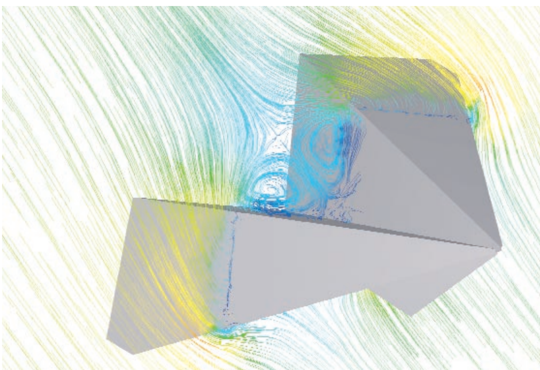
Ivy League

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

The Thayer School of Engineering at Dartmouth College has used FLUENT for many years. Until recently, its primary use was for research and development projects. For the past two years, however, a graduate class on applied computational fluid dynamics has been offered. The course covers the fundamentals of CFD, including numerical methods, turbulence models, mesh generation, and solution strategies. During the class, the students work on a CFD project of their own choosing. The projects have led to a number of interesting CFD results. Examples include modeling the flow through Antarctic firn (compressed snow) layers, heat transfer and phase change in a Southpole hot water drill used to collect micrometeorites (cosmic dust) at different depths, flow in a hydrogen-water separator, free surface flow around boat hulls, shallow water modeling on the continental slope, flow through nozzles, flow in a beaker with a magnetic stirrer, a radial compressor, and airflow around an America's Cup yacht.

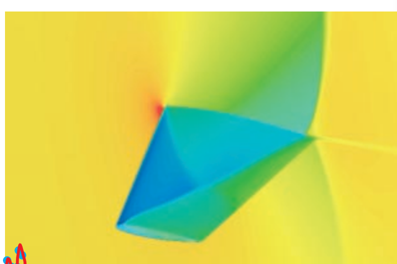


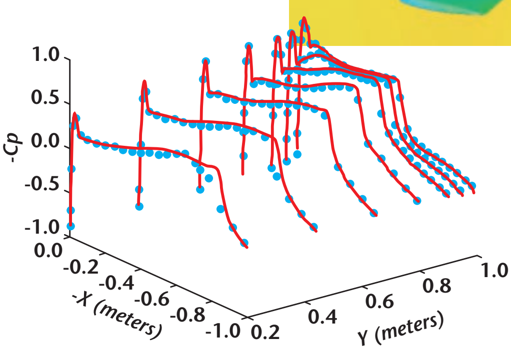
Pathlines illustrate the secondary flow surrounding a paper airplane in flight

Paper Airplane Competition

Students **Kyle Rick** and **Burkhard Lewerich** each folded a paper airplane, one that is commonly used in American classrooms and one that rules the sky in the German Schule. After extensive scientific experimentation, they concluded that the American design flew farther, but its flight was characterized by a destabilizing wobble. The German design, on the other hand, did not fly quite as far, but had a smooth and stable glide. What better way to analyze this than by using FLUENT?! Both designs were meshed in GAMBIT and flow field calculations were performed with FLUENT for a number of different angles of attack. From the flow fields they extracted drag, lift, and the torque on the plane. They then used this information to extract the flight path of each design, reproducing the experimentally found results. After visualizing the flow with pathlines, they found that the additional flap on the German folded design increased the drag (resulting in a shorter flight path), but resulted in a different vortex structure that stabilized the flight.

Contours of density show shocks formed by an ONERA M6 wing flying at Mach 0.93 and a 6 degree angle of attack





FLUENT predictions for the normalized pressure coefficient for the ONERA M6 wing at 0 degree angle of attack are in good agreement with data at several spanwise locations

ONERA Wing Modeling

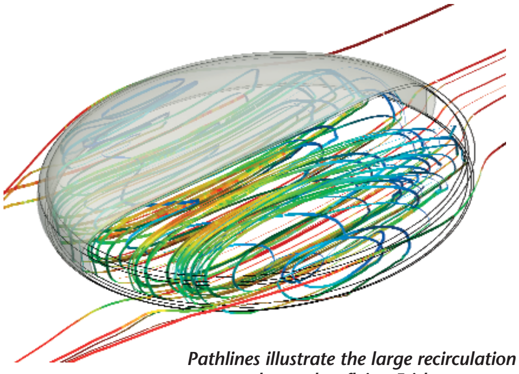
Edward Hopkins took a different perspective to study airplane flight, by modeling the flow around a three-dimensional transonic airfoil, the ONERA M6 wing. He obtained the mesh from NASA and the experimental data for three flight conditions from ONERA (The French National Aerospace Research Establishment). The three conditions were for a zero-degree angle of attack at Mach numbers of 0.7 and 0.92, and a six-degree angle of attack at Mach 0.93. The data consisted of pressure tap data along a cross section of the wings at different spanwise locations. He calculated the flow fields using the coupled implicit solver, second order upwind differencing, and the Spalart-Allmaras turbulence model. FLUENT predictions for the pressure at the upper pressure taps for one set of conditions were found to be in very good agreement with data. Through contours of density, the CFD results were also used to illustrate the various shocks that form at a six-degree angle of attack and a free stream Mach number of 0.93.

CFD Simulations

Baker Library, Dartmouth College

Frisbee® Aerodynamics

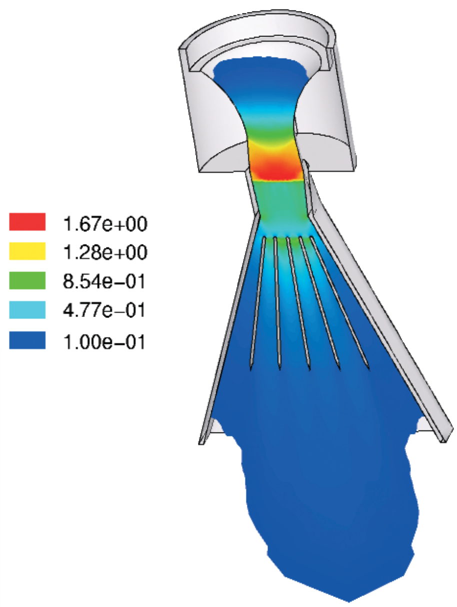
An investigation of another type of flying object was performed was performed by **Ross Gortner**, who modeled the flow around a Frisbee. Ultimate Frisbee, or simply Ultimate, is a fairly recent sport with a long history that dates back to 1871 when the Frisbee Pie Company was founded. Frisbee pies became popular at Yale, and the students developed the habit of playing catch with the empty pie tins. Today, Frisbees are usually made from plastic resins, and the Ultimate Players Association organizes college tournaments in which Dartmouth College teams participate. Ross cut a Frisbee in half so that he would be able to accurately measure the cross-section. He then built the geometry and mesh in GAMBIT and modeled the flow with FLUENT. One interesting aspect of the flow around a Frisbee that was revealed in the simulation is that a vortex forms on the inside of the rotating disk. This recirculation zone may serve to enhance the lift.



Pathlines illustrate the large recirculation zone underneath a flying Frisbee

SAE Formula Racecar Diffuser

A fourth example relates to an exciting application in land transport. **Diana Martin** and **Axel Schmidt** used FLUENT to design a new diffuser for the air intake of Dartmouth College's formula racecar. Formula racecars adhere to a certain formula involving design guideline parameters. They are open-wheeled, open-cockpit, single-seat racecars weighing between 400 and 500 pounds, with an engine less than 600 cubic centimeters in size, and are raced at the annual Formula SAE (Society of Automotive Engineers) racecar competition in Detroit. The rules limit the cross-sectional area of the air intake, so diffusers are used to get the maximum possible airflow to (and horsepower from) the engine. Using the coupled implicit solver, the students compared systems with three different diffuser designs to a system with no diffuser, and selected the system with the highest pressure recovery coefficient. The CFD results, as well as experiments performed earlier, indicated that using a diffuser is a lot better than not using one. The flow in the diffuser is choked, with a peak Mach number of 1.6. Only one diffuser design was successfully tested experimentally, but using CFD, the students tested multiple designs and identified the best one. A diffuser for this year's car was already built, but the new design may be used in next year's model.



Contours of Mach number in one of the simulated diffuser designs shows choked flow

All in all, the graduate CFD course at Dartmouth has been very successful, and the students have shown a great level of creativity in applying CFD to situations that are not commonly studied. ■