Riding on Air with CFX Simulation

Software helps design air cushion vehicle (ACV) lift fans and propellers in less time.

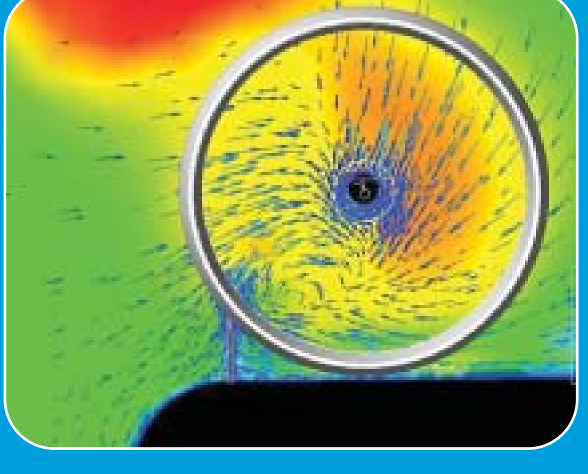
By John Purnell
Senior Engineer
CDI Marine Systems Development Division

CDI Marine Systems Development Division (CDIM-SDD, formerly Band, Lavis, & Associates, Inc.) used computer simulation to design the lift fans and propellers for a proposed large air cushion vehicle (ACV) in a fraction of the time that would have been possible using prior conventional methods. The proposed ACV will be used in transporting cargo and passengers, and is designed to carry twice the payload of the present craft. CDIM-SDD was assigned the task of designing new propellers capable of delivering about twice the power of the present propulsion system. There was also a requirement to design new lift-air supply fans capable of developing additional pressure and flow to support twice the payload.

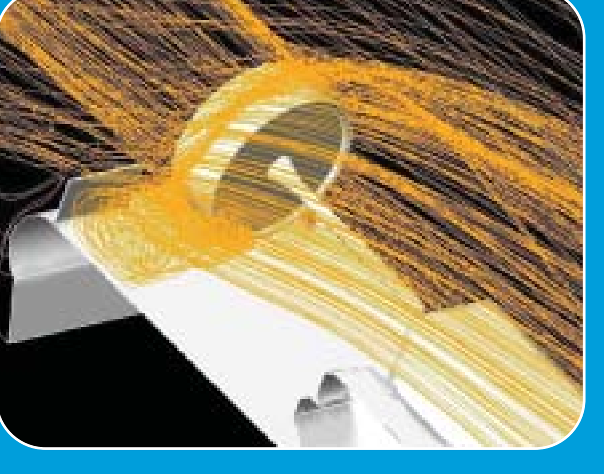
Blade design software was used to generate concept designs for each system. Then CDIM used ANSYS CFX computational fluid dynamics (CFD) software to simulate the performance of each design, including the predictions of horsepower, thrust and inlet velocity profiles for the propellers and predictions of horsepower, flow rate and pressure rise for the lift fans. With the ANSYS CFX program, pretty much any kind of fluid flow analysis can be performed. Basically, if the appropriate geometry of the problem can be set up, the ANSYS CFX analysis can be accomplished. By using a cluster of four 2.6 to 2.8 GHz computers to run problems, running time is improved and larger problems with more detail can be tackled with the ANSYS CFX software, which automatically partitions the problem to run on the multiple computers.

Modeling, Simulation and Redesign

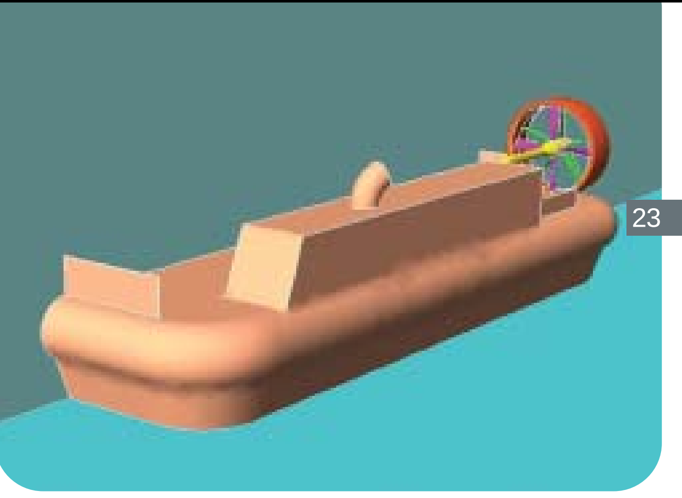
CDIM began by modeling the present product lift fan blades on the existing ACV in the CFX-5 product using CFX-TurboGrid meshing software to mesh the blades. The propeller system design was undertaken following the lift fan system design, which established the total remaining propulsion system power available for the propellers. CFX-TurboGrid software was used to generate the blade passage mesh for both the propeller blade and its stator blade row. CFX-Turbo-Grid enabled CDIM to quickly vary and mesh features (such as the tip gap between the shroud and the



The velocity contours forward of the propeller shroud with tangential velocity vectors shown to indicate the difficult flow field in which the propeller design must operate. High velocity region at top left is caused by the high velocity flow from the forward bow thruster.



Streamline traces of the airflow into and out of the propeller shroud show that inflow comes primarily from well above the deck with additional inflow recirculated from the aft deck tailgate.



CAD model of the air cushion vehicle

propeller blade tip), and to mesh different angles of attack or loadings for the propeller blade by rotating the propeller blade about its center axis.

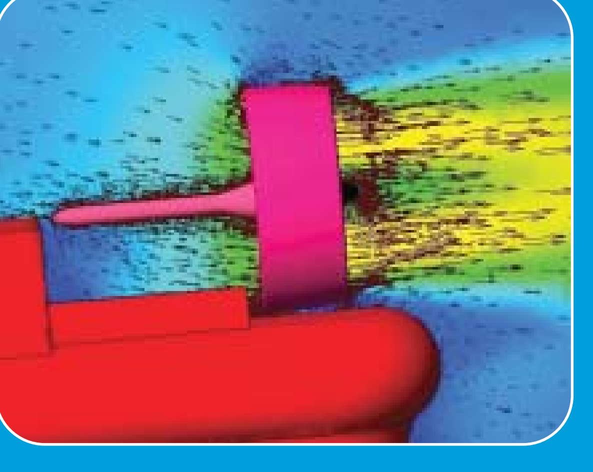
The baseline analysis showed good agreement with physical tests of the existing ACV. So CDIM went to work immediately to create the new design. Data was taken from the baseline ANSYS CFX studies and entered into a blade generation program that allowed control of many other design parameters. This software generated new blade section profiles that were then imported into CFX-TurboGrid for meshing. The fan system modeled in ANSYS CFX included an external area that fed a stationary inlet bellmouth, the rotating fan with blades, and the double exit volute with its two separate and different discharges. A full vehicle analysis using ANSYS CFX provided the inlet velocity profiles, which are critical for accurate predictions of horsepower and thrust for the propeller system.

Optimized Solutions from Iterative Analysis

The results of each ANSYS CFX analysis run were iteratively fed back into the blade design software to produce revised designs. Using this process, CDIM generated scores of potential designs while iterating toward an optimized design solution. This process took orders-of-magnitude less time than would have been required to design, build and test just one design alternative using conventional methods. Finally, CDIM simulated with CFX-5 the wind tunnel used to test the propeller blades. With a good level of accuracy, the simulation was correlated with the results of the subsequent physical testing.

This project demonstrates that ANSYS CFX is an invaluable tool for analyzing system designs. A key advantage is that all components can be analyzed together to view their interface effects and evaluate total system performance. Even simple geometries can become very complex to evaluate when they are tied together. Using an iterative approach between our various design tools, methods and the ANSYS CFX results, CDIM was able to refine the component designs to develop a lift fan and propeller system that efficiently met the ACV requirements in a reasonably short amount of time with minimum physical testing. ■

The distribution of static pressures at the fan rotor exit with the fan mounted inside a volute, which distributes the fan flow.



The vertical propeller centerline cut of velocity contours with velocity vectors indicates the acceleration of flow into the shroud and the distribution of velocities downstream of the propeller shroud.