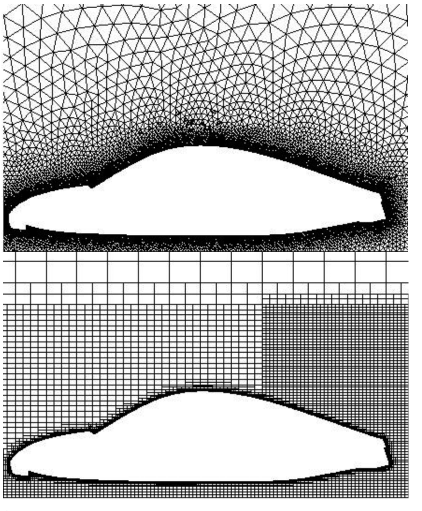
The Immersed Boundary Approach to **Fluid Flow Simulation**

A rapid, reliable and automated approach simulates complex flow fields.

By Chris Wolfe, Senior Product Manager, ANSYS, Inc.

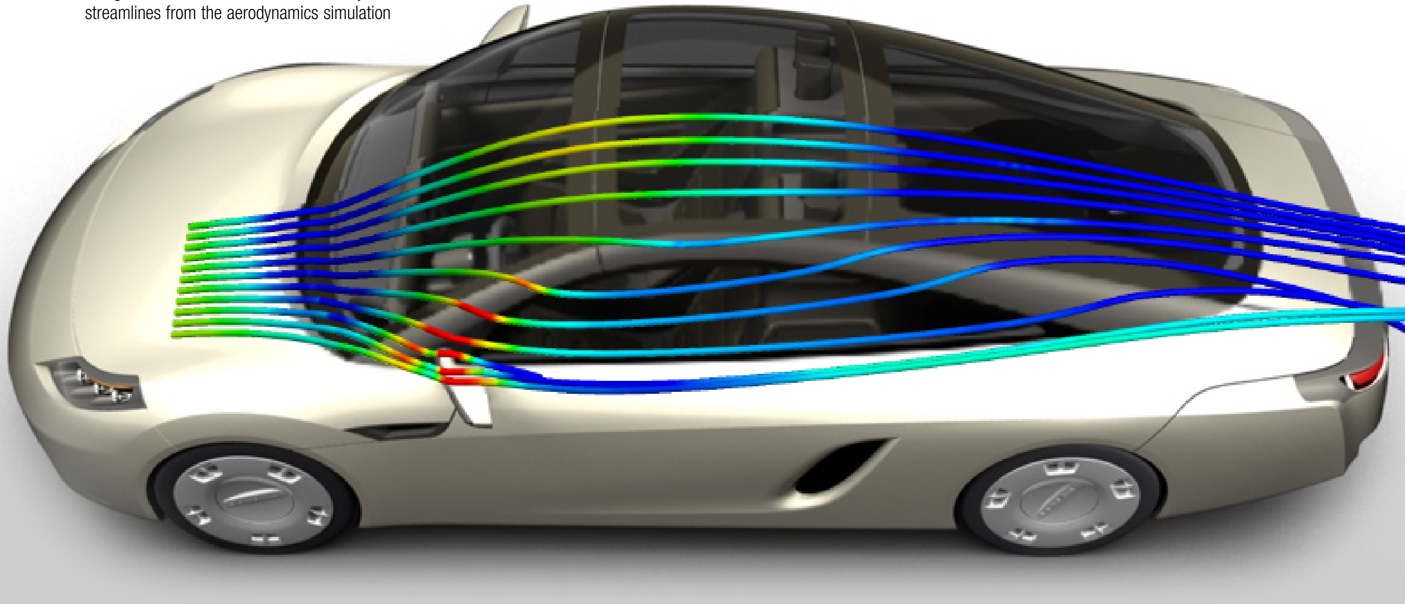
A conventional CFD simulation starts with the transfer of CAD data to a grid-generation package in which a surface mesh and then a volume mesh are generated before the simulation can be set up and solution run. The effort and time required to take geometry from a CAD package and generate a mesh is usually a large portion of the overall human time required for the simulation, and it can be significant. For cases with complex or dirty geometry requiring CAD cleanup, this part of the process may take from 50 percent to 90 percent of the total time required for the simulation. The immersed boundary (IB) method, available through an add-on module for ANSYS FLUENT 12.0 software — jointly developed by ANSYS, Inc. and Cascade Technologies — dramatically reduces the amount of time needed for fluid flow simulations and provides fast results by directly addressing the challenges associated with this step. Since multiple geometries can be analyzed quickly, IB is especially useful as a preliminary design analysis tool for identifying candidates for further higher-fidelity analysis.

The power of the immersed boundary method lies in the meshing technique. Fluid flow simulations using the Immersed Boundary module for ANSYS FLUENT 12.0 software start with the surface data of the simulation geometry in STL format. This CAD geometry does not need to be clean, does not require smooth surfaces or geometry connectivity, and may contain overlapping surfaces, small



Conventional meshing (top) compared to the immersed boundary method (bottom)

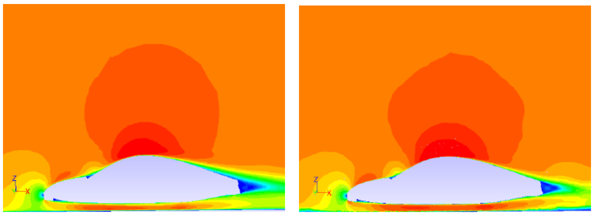
Design of the Loremo vehicle with velocity streamlines from the aerodynamics simulation



holes and even missing parts. The simulation geometry is meshed using automatically generated non-body fitted Cartesian meshes, in which the computational grid does not conform to the geometry. Mesh refinement is carried out automatically, by specifying the resolution on the desired boundaries. Using the IB meshing technique greatly reduces the amount of time spent preparing the geometry for meshing and creating the mesh. It also avoids time-consuming and error-prone CAD-to-CFD geometry conversions and clean-up issues, such as misinterpreted CAD geometrical entities and connectivity.

When creating meshes with the IB module, the immersed boundary mesh cells are not body-fitted and are usually not aligned with the geometry face. Therefore, when using the IB module, the CFD-governing equations are modified in the cells cut by the interface by adding source terms. This allows physical boundary conditions to be enforced through reconstruction schemes that emulate body-fitted behavior in combination with commonly used turbulent wall functions. This first release of IB is fully parallelized and supports the ANSYS FLUENT physical models and boundary conditions needed for modeling low-speed external aerodynamics and automotive front-end airflows.

Essentially, the IB module provides a rapid, automated, preliminary design approach by allowing an adaptively refined high-quality Cartesian mesh to be automatically generated on complicated STL surface representations. The tool offers the ability to save customer time and money by reducing the human effort needed to go from CAD to CFD results. ■



Magnitude of velocity comparison between traditional method (left) and simulation using immersed boundary conditions (right) showing very similar results even though much simulation time was saved. Red indicates higher magnitude of velocity.

References

[1] Cascade Technology, ANSYS, Inc. Immersed Boundary in FLUENT, Technical Documentation.

[2] Mittal R.; Iaccarino G. Immersed Boundary Method. *Annual Review of Fluid Mechanics*. **2005**.

[3] Kalitzin G.; Iaccarino G. Toward Immersed Boundary Simulation of High Reynolds Number Flows. *Center of Turbulence Research, Annual Research Briefs*, **2003**.

[4] Iaccarino G.; Verzicco R. Immersed Boundary Technique for Turbulent Flow Simulations. *Applied Mechanics Review*, **2003**, Volume 56, Number 3.

[5] Jindal, S. et al. The Immersed Boundary CFD Approach for Complex Aerodynamics Flow Predictions. SAE 2007 World Congress, Detroit, Michigan, U.S.A. 2007-01-0109.

[6] Lanfrit, M. A Best Practice Guideline to Handle Automotive External Aerodynamics with FLUENT. *Fluent Inc. Technical Notes*, **2005**, February.

[7] Artiaga-Hahn et al. Customizing FLUENT to Speed Up Aerodynamic Vehicle Development. European Automotive CFD Conference, Frankfurt, Germany, **2005**.

[8] Seibert, W.; Lanfrit, M.; Reese, H.; Hemmer, H. The Immersed Boundary CFD Approach — A Rapid, Reliable and Highly Automated Method to Simulate Complex Flow Fields. *Proceedings of Co-Relation — 7th International Vehicle Aerodynamics Conference*, **2008**.

Rapidly Designing a Low-Fuel Consumption, Low-Emission Vehicle

By Hans Peter Hemmer formerly of Loremo AG, Munich, Germany
Werner Seibert, Industry Solutions Specialist, Marco Lanfrit, Consulting Manager and Hauke Reese, CFD Engineer, ANSYS, Inc.

Vehicles that contribute to preserving the environment don't need to sacrifice cost effectiveness, safety and good looks. These are some of the main objectives of the Loremo (low-resistance mobile) project being conducted in Germany. The basic model of this innovative car will reach a maximum velocity of 160 kilometers per hour, with a fuel consumption of less than 2 liters per 100 kilometers. The pollutant emission of CO_2 will be about 50 grams per kilometer, which is far below the critical value of 130 grams per kilometer proposed by the European Union. At maximum velocity, stability will be an issue, as the car will weigh only 600 kilograms. To meet these project objectives, aerodynamics is one of the key design and optimization factors for developer Loremo AG in Germany. Engineering simulation helped designers to rapidly and reliably develop and optimize the design of the vehicle.

	Conventional Method	Immersed Boundary Method
Geometry Cleanup	16 hours	Not applicable
Surface Mesh	8 hours	Not applicable
Volume Mesh	0.5 hours	0.25 hours
Mesh Count	2,000,000	3,000,000
Memory for Volume Mesh	2 GB (max)	4.5 GB (max)

Statistical comparison between use of a conventional preprocessing method and the immersed boundary method for the Loremo vehicle. All numbers are approximate.

Aerodynamic optimization was conducted using ANSYS FLUENT fluid flow software. Many iterations were required to determine the design that best reduces drag. Aerodynamic simulation had previously been performed using the traditional process of CAD import, surface meshing and then volume meshing before CFD simulation. With the expectation that considerable time could be gained, ANSYS staff working with Loremo engineers tested the immersed boundary approach to mesh the exterior of the vehicle. Unlike traditional methods, the immersed boundary approach does not require a boundary-conforming mesh and, thus, dramatically speeds up the process of grid generation. The process of the CAD cleanup is not necessary because the surfaces are used directly, and the standard surface meshing phase is eliminated.

The overall comparison of results shows good agreement between the conventional simulation strategy and the immersed boundary method, while the reduction in time was significant — less than an hour as opposed to almost 25 hours for the conventional method.

Loremo is scheduled to go into production in 2011.