

Implementation of Immersed Boundary Method for Rapid and Reliable External Flow Simulations in Automotive

Reza Ghias and Ashok Khondge
ANSYS Inc.

Copyright © 2009 SAE International

ABSTRACT

The conventional approach for simulating external flow field over an automobile is to use body fitted (BF) mesh. However, depending on the geometrical complexity of the boundaries, grid generation and grid quality are major issues and take significant time and cost. Moreover, preparing a proper body fitted mesh requires manual intervention which makes it too difficult to automate the process of external aerodynamics simulation.

A different approach is so called Immersed Boundary (IB) method. It provides a high level of flexibility in handling highly complex geometry. It is usually employed in conjunction with body non-conforming Cartesian grid. Thus, grid generation is greatly simplified. This method can tackle flows with complex stationary or moving boundaries with relative ease.

This paper demonstrates the ease of use of the IB method compared to the BF method and possible use of the IB method to automate the external aerodynamics simulations. In present work the results of front-end flow field simulations of a vehicle with complex geometry are evaluated against corresponding conventional body fitted simulations data.

INTRODUCTION

Computational Fluid Dynamics (CFD) simulations are routinely used in automotive industry to model external flow field around the vehicle. Traditionally CFD is used to reduce the number of the wind tunnel experiments, optimize car shapes in terms of aerodynamic drag and lift, and study salient flow features.

The finite volume method is the most popular numerical method in CFD today. This method is mostly used-to-date with body-fitted grids. CFD simulation with body-fitted grids starts with preliminary step of transferring the geometric entities of interest from the CAD environment to grid generation package. The next step is to extract a water-tight volume from the original CAD model, which typically describes the metal, or solid, components. After water-tight volume is extracted in the grid generator, a surface mesh is created on the all boundaries of volume. This surface mesh will form a base for creating body-fitted volume mesh for CFD computations.

CFD simulations are used to support vehicle aerodynamic development throughout the vehicle design process. Early models may often be simplified with closed cooling openings and sealed engine bays. Later models are more complex with open cooling, detailed

engine compartments, and underbody components. For geometries with such complexity, the pre-processing step typically accounts for large portion of time required to perform entire CFD simulation. Unstructured grids, based on tetrahedral elements, can be generated in an automatic environment, thus reducing the associated time and cost. Unfortunately, the computation of wall-bounded flows poses strict constraints on the mesh quality and limits the applicability of these meshes. Based on the complexity of model the pre-processing step can easily take 50-90% of the time required to perform the entire project, as all of the other steps in the simulation process can be automated quite easily, as shown in [1].

An alternative approach to handle very complex geometry configurations is the Immersed Boundary (IB) method. This method uses automatically generated Cartesian meshes, where the computational grid does not conform to the geometry. The effect of the real geometry configuration on the flow is considered by introducing an appropriate treatment in the CFD solver using source terms in the boundary cells. As the immersed boundary technique is employed in conjunction with a body non-conforming Cartesian grid, the grid generation is greatly simplified and this method can tackle flows with complex stationary or moving boundaries with relative ease [2].

The IB method has been employed and evaluated for a wide range of basic geometries in recent years in academia. The need of cutting cost and reducing time in design process has made this method attractive enough to be considered to use in automotive industry commercially [3] because real-life applications of CFD in the automotive industry typically involve extremely complex geometries.

In the present work, we first describe the theory of immersed boundary method briefly. Then a commercially implementation of this technology which is being collaboratively developed by Cascade Technologies and ANSYS Inc., is implemented to simulate front-end simulation for a sedan model. This production sedan model has all the details including complex underhood and underbody components. Later we present the results of external flow simulation using body-fitted grid and IB method.

IMPLEMENTATION AND THEORY OF IMMERSED BOUNDARY METHOD

The present implementation of Immersed Boundary Method relies on the use of Cartesian meshes and modified boundary condition treatment at the cells crossed by physical boundaries. Figure-1 shows the

three different approaches, which are based on Cartesian meshes using non body fitted grids.

A uniform mesh resolution is not sufficient to account for all salient flow features in a CFD simulation. The IB implementation in FLUENT automatically refines meshes in near wall areas, based on a few parameters given by the user. Such local mesh refinement is carried out in an automatic fashion by specifying the desired resolution on each of the physical boundaries. For external aerodynamics simulation this is of special interest for wake region and vehicle parts exposed to the flow, like side view mirrors. Figure-2 illustrates the local mesh refinement near the front-end of the sedan used in the present study.

The schematic representation of grid refinement using adaptive Cartesian grids is shown in Figure-3. The conventional OCTREE approach (left) is based on a splitting of coarse cells (father) to generate smaller elements (kids); this introduces hanging nodes and complex connectivity between cells. In addition, it is complicated to extend this to a non-isotropic environment where 2, 4 (or 8 in 3D) kids can be present at each level in the refinement step. A fully unstructured mesh approach (right) produces polygonal (polyhedral in 3D) elements, that always has proper connectivity between neighbors. This method is more complex to handle but provides flexibility in accommodating anisotropic refinement. The mesh refinement is carried out in an automatic fashion by specifying the desired resolution on each of the physical boundaries. Cells in a uniform initial grid are typically decomposed into smaller cells: from one "father" cell to several smaller "kid" cells. The present method can accommodate anisotropic refinement (each father cell can be split selectively in one or two directions), which results in considerable savings in overall mesh size when the boundaries are largely aligned with the Cartesian direction. In addition, the grid generation process does not introduce the father-kid relationship between cells and the corresponding hanging nodes, but it produces a truly unstructured grid, as illustrated in Figure-4. These meshes are treated in FLUENT as polyhedral cells.

The Immersed Boundary module in FLUENT consists of three major parts:

- 1) TOMMIE - A grid generator that produces the Cartesian grid and the information required to locate the physical boundaries.
- 2) A set of procedures and boundary treatments to be included in the solver algorithms to create the effect of boundary walls.
- 3) A post-processing module that allows visualization of the results on the physical boundaries.

In summary, the procedure follows four steps:

- 1) The geometry of interest is imported in the system and a Cartesian mesh is generated according to the desired resolution specified by the user.
- 2) The cells crossing the immersed boundary are identified and geometrical information that relates the boundary location with the mesh elements is computed.
- 3) The grid and the additional geometrical information are imported into the flow solver.
- 4) "Solving as usual" either in serial or in parallel.

GEOMETRICAL DEFINITION

The immersed objects can be described using CAD primitives directly, thus eliminating completely the need for CAD to CFD translations. The widely used Stereo-Lithography (STL) format is employed within the IB implementation in FLUENT. The STL representation of surface is a collection of unconnected triangles of sizes inversely proportional to the local curvature of the original surface. This format is already the standard for the Rapid Prototyping community and all CAD systems have the ability to export given surface in STL format automatically.

Multiple parts can be imported. Even if they intersect each other it does not create any issues in generating the mesh. The only requirement for the object description is that the given surface must be a closed manifold, like for rapid prototyping tools. The surfaces do not need to be connected.

GRID GENERATION

A Graphics User Interface (GUI) driven mesh generation tool called TOMMIE (Tired of Meshing) has been developed in the context of the IB method implementation into the FLUENT code. It creates a hexahedral mesh with isotropic or anisotropic mesh refinement based on users input.

TOMMIE generates the computational grid starting from a uniform hexahedral mesh, which is successively refined to meet the user specified resolution normal and tangential to boundaries. User defined refinement areas can be defined by specifying their location, shape and desired resolution. A refined hexahedral mesh is created, which includes fluid and solid parts. The cells placed in the fluid area are marked by a material point (physical location) defined by users. The solid cells are deleted. The cells crossing the immersed surfaces are identified as interface cells which later will require appropriate boundary condition treatment.

The boundary conditions are constructed from the knowledge of the local immersed surface geometrical information. The result is a stair-step representation of the geometry of interest as shown in Figure-4. Each face of this stair-step boundary is associated with one and only one location on the original STL surface. So, by definition, the line connecting the cell centroid and the stair-step face centroid contains an intersection with the original STL surface.

The usage of input files for TOMMIE makes the tool quite flexible. This is illustrated in Figure-5 where for 4 different sphere-sphere configuration meshes have been created, using the same TOMMIE input file. Based on the position of the two geometrical entities to each other, different fluid domains are created. This is even true, if the same boundaries of the entities do overlap or one component is fully inside each other. The intersection has not to be computed a priori. This allows the use of complex CAD assemblies directly in this context and also provides a flexible framework for design variations.

The last step of the grid generation process is the definition of an immersed boundary representation that is compatible with the volume grid generated. This surface representation is required for post processing. It is not possible to use STL representation directly as it may contain triangles of size widely different from the grid element size in its vicinity. The surface is instead "reconstructed" from the stair-step representation already available. A projection operation is used to reposition each face of the stair-step surface onto the true surface. The end result of this operation is illustrated in Figure-6.

ENFORCING BOUNDARY CONDITION

The objective is to specify boundary conditions on the stair-step, boundaries that are representative of the "true" boundary conditions for all the relevant quantities. This can be accomplished by interpolating the computed values inside the flow domain and the desired assigned values on the true boundary into the target location (the stair-step face centroid). Details of the interpolation (order of accuracy, support stencil, etc.) do not seem to be extremely important as the effect on the overall solution accuracy is limited. It is important though that physical conservation laws are respected. The implementation of wall models for turbulent flow calculations requires an additional modification.

The current interpolation scheme used for enforcing Dirichlet-type boundary conditions is based on a least square interpolation (LSI) procedure. For each quantity ϕ the boundary condition is obtained as:

$$\phi_{IB} = \sum_{nbr=1}^{N_F} \omega_{nbr} \phi_{nbr} + \sum_{ss=1}^{N_B} \omega_{ss} \phi_{ss} \quad (1)$$

where IB is the desired face value at the stair-step boundaries and N_F and N_B are the fluid neighbors and the cell-faces that belong to the immersed boundary where “true” boundary values are specified, respectively. The weights ω are only a function of geometrical distances between the immersed surface and the face and cell centroids, and are computed within Tommie. For vector quantities (e.g. the velocity) an additional step is required to enforce conservation at the stair-step boundary. Mass conservation (for an incompressible fluid) can be obtained by enforcing that

$$\vec{v}_{IB} \cdot \hat{n} = 0 \quad (2)$$

At each of the stair-step faces ($\hat{n}$ is the normal to the stair-step face, not the true immersed surface). This can be accomplished by rewriting the interpolation step as a predictor/corrector procedure

$$\begin{aligned} \vec{v}_{IB}^* &= \sum_{nbr=1}^{N_F} \omega_{nbr} \vec{v}_{nbr} + \sum_{ss=1}^{N_B} \omega_{ss} \vec{v}_{ss} \\ \lambda &= \frac{1}{N_{IB}} \sum_{IB=1}^{N_{IB}} (\vec{v}_{IB}^* \cdot \hat{n}) \\ \vec{v} &= \vec{v}_{IB}^* - \lambda (\vec{v}_{IB}^* \cdot \hat{n}) \end{aligned} \quad (3)$$

Strict local mass conservation is enforced by computing the correction factor λ on a face-by-face basis and global conservation is equivalent to computing λ as a sum over all the stair-step faces. Our experience has been that extending the sum for the correction factor to a few faces is sufficient to achieve second order accuracy [4].

The application of the immersed boundary techniques for Reynolds-Averaged Navier-Stokes applications requires the introduction of boundary conditions for the turbulent quantities and the ability to define a wall model (wall functions). Direct enforcement of no-slip wall boundary conditions is appropriate only for properly resolved near wall regions ($y^+ \sim 1$). An alternative treatment based on a wall model is derived from a known functional dependency of the tangential velocity and the turbulent quantities (including the eddy viscosity). The wall model concept is particularly important in this context because it is difficult to achieve high resolution within boundary layers (unless very large meshes are considered). The blending of the immersed

boundary treatment with a conventional wall function approach is discussed in detail in [5].

In FLUENT, the IB formulation relies on the methods used for body-fitted meshes so that almost every turbulent model and their variants are supported. In addition, to enforce a boundary value for the velocity and the turbulent scalar, source terms have been added to the cells directly connected to the immersed boundaries to ensure that the current wall friction is considered [6].

PROBLEM DEFINITION

The sedan model studied in the present work is a full scale production car model. This model includes all the details such as underhood and underbody components, power train components. Figure-7 shows the rendition of the sedan geometry used in present study. Figure-8 shows the sedan model in wind-tunnel. Figure-9 shows some of the complex parts like engine assembly, cooling package assembly of underhood components.

In the present work the CFD simulation of sedan model using conventional body-fitted mesh method is compared with the immersed boundary approach. The elegance and ease of use of Immersed Boundary approach is demonstrated. The front end external flow simulation is carried out using both body-fitted mesh method and Immersed Boundary approach and front-end flow simulation results are compared.

MESH GENERATION PROCEDURE

In this section the mesh generation procedures for sedan model using conventional body-fitted (BF) mesh method and immersed boundary (IB) approaches are described and compared.

BODY-FITTED APPROACH

The general requirements of a mesh used in finite volume method are discussed in detail in literature [7, 8, and 9]. For front end flow simulation the specific local refinement is required near front end, grid in front of radiator and condenser is required to resolve the flow features. In the present study TGrid 5 [9] is used to create body fitted (BF) surface mesh. The surface wrapper technology in TGrid is used to create the surface mesh.

TGrid surface wrapper will start with CAD models in STL format. The CAD to STL conversion can be done in most of the CAD software (e.g. Pro-E or CATIA). The wrapping process is based on the Cartesian grid (or overlay grid) approach.

The overall process of creating surface mesh using TGrid 5 wrapper is as follows;

1. A coarse Cartesian grid is overlaid on the input geometries of various components of Sedan model. This Cartesian grid is used to automatically clean the input geometry and to create the water-tight representation.
2. The intersection between the Cartesian grid and the input geometry is calculated and TGrid identifies and marks the intersecting cells.
3. A watertight faceted representation is created along the boundary of intersecting cells.
4. The nodes on this faceted representation are projected on the faces and feature edges of the input geometry which then results in a wrapper surface closely representing the input geometry.
5. The wrapper surface quality is improved by post-wrapping operations such as smoothing and swapping.

It is possible to automate the process of wrapping in TGrid for complex geometries however this will require significant expertise. In the present study the surface mesh and volume mesh for body fitted (BF) case was generated using semi-automated way using TGrid journal scripts.

The body fitted (BF) mesh was generated using Linux machine of 32 GB RAM with peak memory requirement of 12 GB. TGrid wrapper initialization process requires significant RAM and it requires high RAM machines. The total cell count was 12.2 million cells. It took approximately 4 days to create the body fitted mesh. Figure-10 and 11 show the body-fitted surface mesh on some of the parts of the sedan model and volume mesh on cut plane respectively.

IMMERSED BOUNDARY APPROACH

Immersed Boundary method mesh generation is far more automatic and therefore considerably less time consuming than corresponding body-fitted mesh generation method described in previous section. For IB mesh generation tool, TOMMIE, the input geometry files are required in STL format. For the present study various components of Sedan geometry were available in STL format. Most of the CAD software (e.g. Pro-E, CATIA) have features to convert geometric data into STL format. It is important to mention that no clean up has been performed on generating the IB mesh. In other words, the individual STL representations of various components of Sedan geometry can be inter-penetrating. TOMMIE can automatically handle such multiple overlapping bodies. This can save considerable time and effort depending on the quality of CAD data and complexity of the geometry. For present study

separate STL files for various components like body, engine assembly, tires etc were used. The desired surface mesh resolution can be applied for these individual components.

TOMMIE is embedded in Fluent by using additional modules that need to be installed for IB method. Thus mesh generation can be easily done in Fluent software by using a Graphical User Interface (GUI). It is also possible to use text editor to write the TOMMIE input file and read it into Fluent or run TOMMIE in background option to create an IB mesh.

The TOMMIE input file used for creating IB mesh is shown in Figure-12. This script instructs TOMMIE to generate the required mesh. First 23 lines of script address the input STL files for various components like body, engine assembly, mirror etc. Specific format (STL_INTERIOR) is used to assign any internal surface as interior zone as described on line number 9 and 10 for CAC (charge air cooler) inlet and outlet internal surfaces, line number 12 and 13 for radiator inlet and outlet internal surfaces, line number 15 and 16 for condenser inlet and outlet internal surface. Fan is modeled using planer fan boundary condition. The interior surfaces for planer fan models are described on line number 18 and 19 of the TOMMIE input script. Line 24 of this script describes the computational domain extent. Line number 25, 26, 27 and 28 describes various fluid zone. To apply porous media conditions for radiator and condenser, they are defined as separate fluid zones. IB mesh smoothing and refinement parameters are described on line number 29 and 30 in the script. Finally last line provides the name of the generated IB mesh file.

TOMMIE generates two files first the case (*.cas) file and second the ib file (*.ib). The former is a regular FLUENT case file while the later includes geometrical information that relates the boundary location with the mesh elements in FLUENT 6.3 [10] as a solver.

Mesh generation was performed using parallel mesh generation capability in TOMMIE. The mesh was generated using 8 CPUs, each with 2 GB RAM and 2.4 GHz CPU speed. This mesh was generated in 2 hours. The total cell count was 13.5 million cells. This mesh generation time is over 48 times smaller than that required for creating the body-fitted mesh. In the present study the mesh sizes in the normal and tangential directions on the various components of Sedan are similar to the corresponding body-fitted sizes. Figure-13 and 14 shows the projected IB surface mesh on some parts of the sedan model and IB volume mesh on cut plane respectively. Table-1 shows the comparison of mesh generation process for BF and IB method.

NUMERICAL SIMULATIONS

FLUENT V6.3 with Realizable $K-\varepsilon$ turbulent model was used as a solver for both BF and IB simulations. Flow field was assumed steady state incompressible and a second order implicit finite volume scheme was used to solve three dimensional Reynolds Averaged Navier-Stokes (RANS) equations. Table-2 summarizes the solver set up for front end flow simulations. The boundary conditions used in simulations were listed in Table-3. A constant velocity boundary condition was chosen for the inlet in order to easily replicate tunnel inlet velocities observed in the experiments. Since we assume the flow to be incompressible, an arbitrary value of pressure can be assigned to the pressure outlet without any effect on the flow field, so it is kept at 0 Pa (gage). Tunnel sides and top are given the symmetry boundary condition. The road and vehicle surfaces are all given no-slip boundary conditions in order to closely replicate the actual wind tunnel used in the experiments. The fan is modeled using planer fan model. Radiator and condensers are modeled using porous media model.

For BF simulation, the solution was converged with 1st order discretization schemes first. Then the discretization schemes were changed to their final settings as in Table-2. The solution was iterated until the convergence. For IB simulation, the discretization schemes were set to second order from the beginning of the simulation which is another advantage of using the IB method.

RESULTS AND DISCUSSION

Figure-15 shows the velocity magnitude contours on $y=0$ plane. It is worth to mention that since the main purpose of this simulation was on finding the flow field and mass flow rates in underhood, therefore we have not used a fine mesh in other regions as can be seen in figure-16. In order to compute the drag force in external flow simulation accurately, it is necessary to employ mesh refinement in the wake of the geometry. This has been addressed in previous paper by authors in detailed [11]. As it was mentioned before, we have not done any clean up or water tightening process. So we expect to have expected possible leakages. But mass flow results do not show any major discrepancy due to these leakages. Figure-17 and 18 show the velocity contours near front end on $y=0$ and $x=1.75$ m planes respectively in which the velocity contours are in good match compare to corresponding BF results. The lower limit of the velocity in color map has been set up to 1.0 m/s in order to show the source of the small leakages in IB method for this simulation in compare to water tight geometry in BF simulation.

Figure-19 shows the static pressure contours on $y=0$ plane. Due to the leakage into the cabin and engine, the pressure contours exist in these locations. Figure-20 and 21 show the static pressure contours near front end on $y=0$ and $x=1.75$ m planes respectively where both IB and BF results are comparable.

Front end velocity vectors on $y=0$ plane is shown in Figure-22. Comparison of static pressure plots across porous zones is presented in Figure-23. Figure-24 shows a comparison of static pressure on the upper part of the sedan on $y=0$. The results are reasonably match in front part of the sedan but due to the coarseness of the mesh in the back of the sedan, the IB and BF results are not quite match on this region. Also it is worth to mention that there are extra points in IB present the pressure inside the cabin and underhood since only one STL geometry has been assigned for the whole body.

Figure-25 and 26 show comparison of pressure contours on inlet and outlet of condenser and radiator respectively.

Table 4 presents the mass flow rates of the air through the radiator and condenser. The IB results are within 1% difference compare to corresponding BF mass flow rates which is quite reasonable.

The front end flow field using path lines are compared for both BF and IB simulations and presented in Figure-27. There are similar trends in both IB and BF pathlines.

In overall, the results in terms of static pressure and velocity contours match very well. Also there is a good match of trend for plot of pressure across the porous media. However IB method slightly under predicts the pressure values across the porous media.

CONCLUSION

Implementation of Immersed Boundary method to simulate external front end flow over a sedan model was studied in this paper. The comparison is made between the simulation process used in the conventional body fitted mesh method and the Immersed Boundary method. Immersed Boundary method can substantially reduce the preprocessing time and it alleviates the need of high RAM machine for preparing big meshes as IB mesh generator is fully parallelized. The results of IB method simulation were in acceptable match compared with corresponding body fitted simulations. The effect of the mesh needs to be investigated in future work.

REFERENCES

- [1] Silvestre Artiaga Hahn, Nicolas Kruse, Frank Werner, "Customizing FLUENT to Speed-up Aerodynamic Vehicle Development", EACC, 2005
- [2] Ghias R., Mittal R., Dong H., "A sharp interface immersed boundary method for compressible viscous flows", Journal of Computational Physics, Vol. 225 pp. 528-553, 2007.
- [3] Jindal S. et al. "The Immersed Boundary CFD Approach for Complex Aerodynamics Flow Predictions" SAE 2007 World Congress, Detroit, Michigan 2007-01-0109 (2007)
- [4] Iaccarino and Ham, "Automatic Mesh Generation for LES in Complex Geometries", Annual Research Brief, Center for Turbulence Research, 2005.
- [5] Kalitzin and Iaccarino, "Towards Immersed Boundary Simulations of High Reynolds Number Flows", Annual Research Brief, Center for Turbulence Research, 2003.
- [6] FLUENT-IB Documentation, ANSYS Inc., (2008)
- [7] Patankar, S.V., "Numerical Heat Transfer and Fluid Flow", Hemisphere Publishing Corp., Washington D.C. (1980)
- [8] Mathur, S.R. and Murthy, J.Y., "A pressure-based method for unstructured meshes", Numerical Heat Transfer, Vol.31, pp. 195-215 (1997).
- [9] TGrid 5.0, User's Guide, ANSYS Inc., Lebanon, NH (200).
- [10] Fluent 6.3 User's Guide, ANSYS Inc., Lebanon, NH (2006).
- [11] R. Ghias, S. Sovani, and A. Khondge, "Flow Simulation around a Generic Ground Transportation System: Using Immersed Boundary Method", SAE paper 2008-01-2601.

CONTACT

Reza Ghias, D.Sc., Consulting Engineer, ANSYS Inc.,
3005 Boardwalk, Suit 100, Ann Arbor, MI 48108

Email: reza.ghias@ansys.com

Phone: (734)-213-1248 Fax: (734) 213-0147