

Rapid Meshing for CFD Simulations of Vehicle Aerodynamics

Keyur Kanade, Ashok Khondge, Ibrahim Yavuz,
Padmesh Mandloi, Hamid Ghazialam, Sandeep Sovani and Laz Foley
ANSYS, Inc.

Robert Lietz
Ford Motor Company

Copyright © 2009 SAE International

ABSTRACT

To-date the primary challenge in conducting aerodynamic CFD simulations of actual vehicles with realistically complex geometry has been the construction of a computational mesh. The CAD-to-Mesh processes used to-date have been laborious, often requiring many weeks of engineering time. In this paper we present a new technique to greatly expedite the CAD-to-Mesh process. The fundamentals of this technique are discussed followed by case studies that show that this technique can reduce the engineering time required for the CAD-to-Mesh process to just a few hours.

INTRODUCTION

As a primary input any Computational Fluid Dynamics (CFD) simulation requires a computational mesh representing the flow region that is to be studied. This requirement exists regardless of the numerical formulation method chosen for solving flow equations such as, the Finite Volume Method (FVM)^[1], the Finite Element Method (FEM)^[2], or the Lattice-Boltzman Method (LBM)^[3]. Further, irrespective of the numerical formulation method chosen, a CFD computational mesh needs to satisfy the following basic requirements –

- A. Accurate representation of domain boundaries
- B. Variable mesh resolution throughout the domain that puts more focus on important regions and reduces numerical error
- C. Quality of discretized elements

While a CFD mesh prepared for any of the numerical formulation methods has to meet these broad requirements, some methods have more stringent bounds on some of these criteria than others. In particular, the traditional implementation of FVM involving solution of the Navier-Stokes and related conservation equations, has especially stringent requirements for the computational mesh in terms of mesh resolution and quality of discretized elements. However, once a mesh with these stringent requirements is prepared, the FVM provides a robust, fast, and accurate solution for a very wide range of flow, thermal and related problems, thus making it the most widely used and popular numerical formulation today.

These requirements are particularly difficult to meet while creating meshes for CFD simulations of automobile exterior flow for applications such as aerodynamics and underhood thermal management. The geometric complexity of a realistic automobile that includes all major components and sub-assemblies such as exterior surfaces, mirrors, grills, powertrain, transmission, suspensions, and others, makes it highly challenging to accurately represent the CFD domain boundaries and to achieve strategic local mesh refinement while ensuring high quality for all individual mesh elements. Thus, mesh generation for CFD simulations of automobile aerodynamics and underhood thermal management is thought to be the “grand challenge” for CFD mesh generation methods and algorithms.

A CFD mesh consists of two parts –

- Surface mesh
- Volume mesh

The surface mesh is a faceted representation of the domain boundaries. Since a CFD computational domain is essentially a finite region of space, its entire boundary is a “water-tight” closed surface. The surface mesh represents this closed surface in the form of a collection of adjacent and tightly connected planar shapes, such as triangles, quadrilaterals or polygons.

The volume mesh is a collection of 3D geometric shapes such as tetrahedra (tets), hexahedra (hexes), prisms, pyramids, and polyhedra, that completely fill-up the region enclosed by the surface mesh. Each individual shape in this mesh serves as a finite volume in the Finite Volume Method (FVM) or a finite element in the Finite Element Method.

The CFD mesh generation process consists of two steps, surface mesh generation followed by volume mesh generation. In the first step a surface mesh is created starting from the CFD domain boundary geometry represented in a basic geometric format such as a native CAD file format (for example, CATIA®, ProEngineer®, UniGraphics®), or a neutral file format (for example, Parasolid, STEP, ACIS, IGES) or a raw faceted format (for example, STL). The second step is to fill-up the volume enclosed by the surface mesh with volume elements.

Of these two steps, the surface mesh generation step has remained a challenge till recently. For the volume mesh generation step, highly automated and robust techniques such as the Delauney Method^[4] have been developed and fine-tuned in the past two decades and are in popular use today. Such methods work robustly and automatically provided that the underlying surface mesh meets requisite quality criteria. Thus, an important task in surface mesh generation is that of meeting these quality criteria.

The primary challenge in the surface mesh generation step has been the lack of an algorithm that can automatically and robustly generate a surface mesh while meeting the three broad requirements mentioned earlier – (A) accurate boundary representation, (B) variable mesh resolution, and (C) element quality. In lieu of such an algorithm the surface mesh generation step for realistic automobile geometries has been fairly laborious requiring several man-weeks or man-months to complete^[5].

In the past two years there has been major progress in filling this gap. Several independent groups^[6, 7, 8, 9] have been working on developing and implementing an algorithm referred to as the “surface wrapper” that is capable of dramatically automating the surface mesh

creation process. Our work shows that this algorithm can reduce the surface mesh generation effort from several man-weeks to a few man-days or even a few man-hours.

In this paper we describe the surface wrapper method and present observations made while using this method for generating meshes for aerodynamics and thermal management simulations of realistic vehicles. The work reported in this paper was conducted using the surface wrapper and other meshing tools in the commercial software named TGrid5.0^[6].

RAPID MESHING METHODOLOGY

SURFACE MESH

The surface meshing step starts with geometric representation of the domain boundaries in basic CAD, CAD-neutral, or raw faceted format. For an aerodynamics or thermal management simulation, this includes the wind tunnel walls, road, inlet, outlet, and the entire vehicle surface including all details such as exterior surfaces, mirrors, grills, powertrain, transmission, suspensions, and others.

As a first step in performing a CFD simulation an analyst collects CAD models of vehicle parts and sub-assemblies from a central database and puts together a geometric representation of the vehicle. Such a geometric representation typically has numerous flaws, as shown in Figure 1, that cause problems in the surface mesh generation process. Typical problems are:

- Small gaps
 - Tolerance issues
 - Parts missing
 - Translation issues
- Thin objects
- Zero thickness surfaces
- Overlaps
- Intersections
- Proximity
- Small details
- Un-wetted surfaces
- External & internal details

A CFD domain is essentially a finite region of space and therefore has one “watertight” closed boundary. The above flaws in the input geometric data set cause problems because they either open-up or disrupt the closed boundary surface (gaps, overlaps, intersections, etc), or provide redundant geometric information (small details, internal un-wetted surfaces, etc.)

The mesh generation task is essentially to start from such a problematic geometric representation of the computational domain, and achieve a volume mesh enclosed by a watertight closed boundary as depicted in Figure 2.

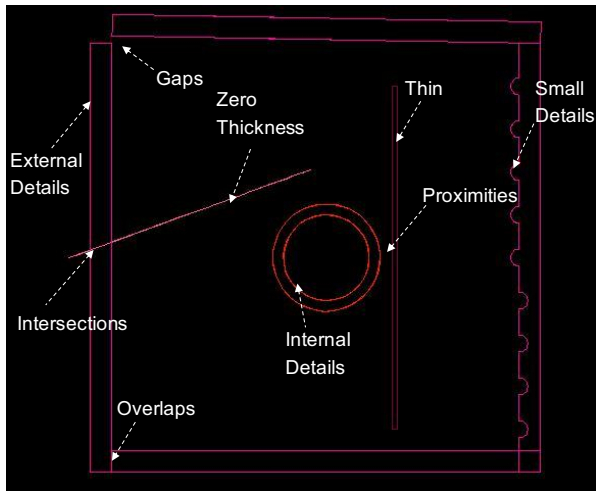


Figure 1. Typical problems in an input geometry data set.

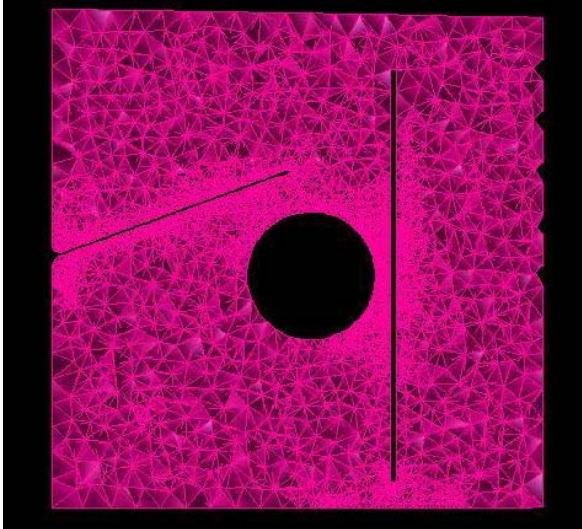


Figure 2. Cross-section of the final volume mesh.

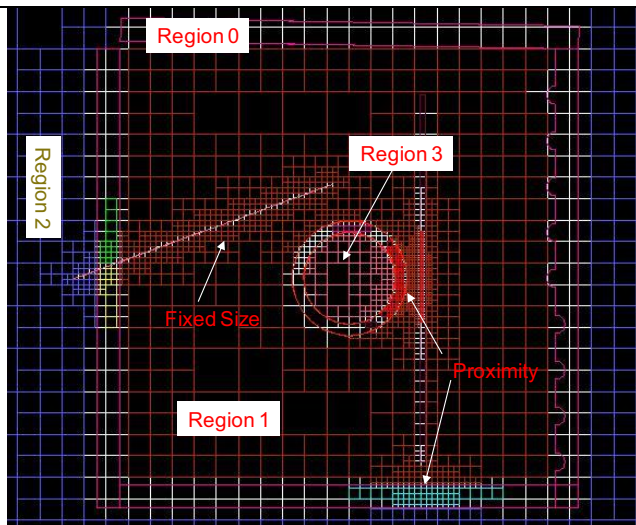


Figure 3. Initial Cartesian mesh.

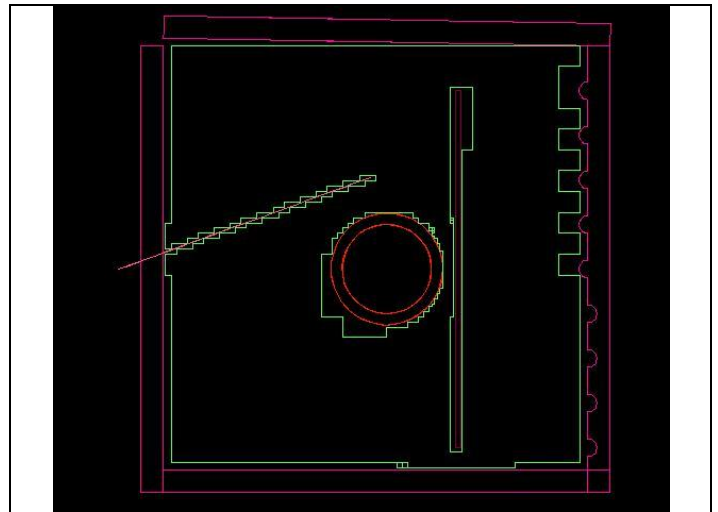


Figure 4. Extracted interface.

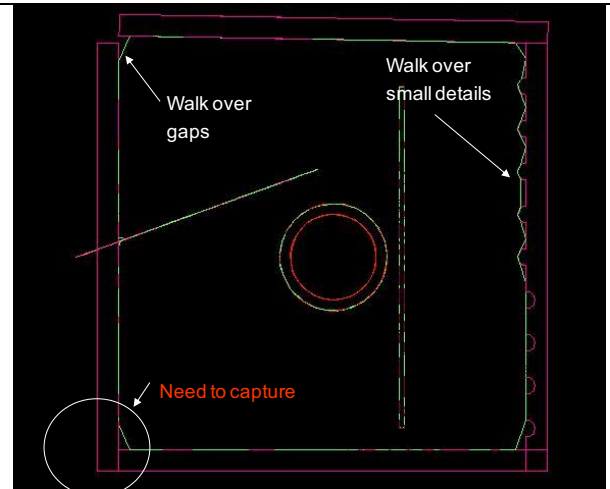


Figure 5. Initial wrap surface.

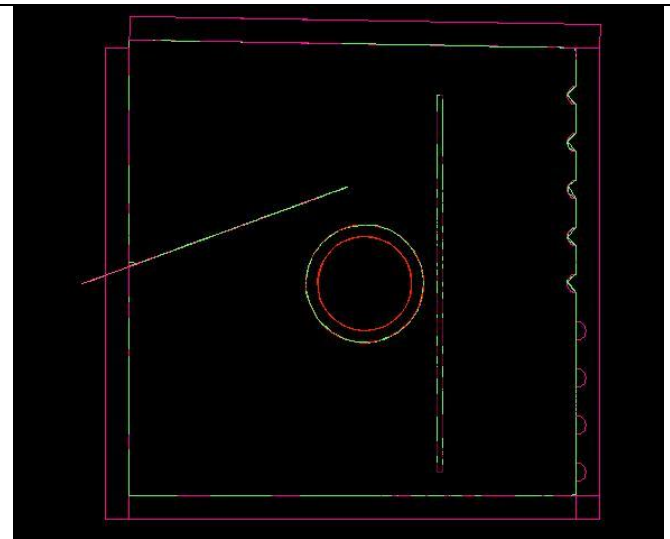


Figure 6. Wrap surface after imprinting feature lines.

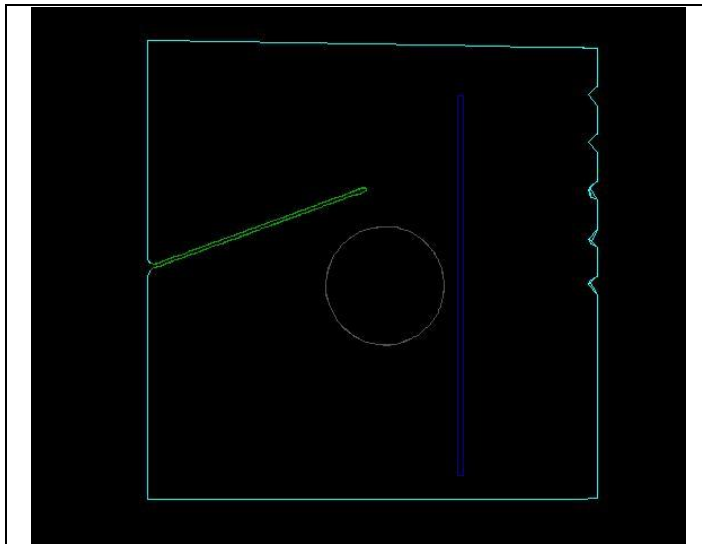


Figure 7. Final surface mesh created by the wrapper.

In the past, in lieu of automated surface meshing tools, such geometric problems were corrected by the analyst by visual inspection of each problem and corrective action. For example, an unwanted gap would be removed by manually creating a surface to span the gap, or any unwanted details would be manually deleted. In a typical geometry data set of a realistic vehicle there are thousands of such flaws that need correction, therefore, in the past the surface mesh generation process was laborious requiring several man-weeks or man-months.

The surface wrapping technique is designed to overcome these problems in a largely automatic way. The various steps implemented by the surface wrapper are described below.

Size Assignment – First the user specifies the mesh resolution needed on different parts of the vehicle depending on the intent of the analysis. For example, for Front-End Flow Analysis, the flow features near the front grill and the cooling package need to be captured with best possible accuracy, therefore a fine mesh resolution specified in this area. However, in this application it is not necessary to capture the vehicle wake accurately, therefore a relatively coarse mesh resolution is provided on the rear end of the vehicle. Unwanted minor details in the original geometry can also be overridden by assigning a mesh size greater than the size of these details in the general area of the details. Sub-algorithms that automatically increase or decrease mesh resolution based on curvature and proximity between parts can also be used to ensure proper grid resolution in regions of prominent flow curvature and in narrow flow passages.

Initialization – In this stage the wrapper algorithm first creates a uniform Cartesian mesh that completely engulfs the original geometry, as well as passes through it. Each cell in the uniform Cartesian mesh is a cube with

side length equal to a pre-specified default value. The assigned sizes, i.e. user-specified mesh resolution, and proximity and curvature based refinement functions are used to refine the Cartesian mesh in specific regions. The hanging-node refinement method is used. This initial Cartesian mesh is shown in Figure 3. Notice that the mesh size is larger than the size of the unwanted details on the right side of the figure.

Once the Cartesian mesh is ready, all the cells that intersect with the original geometry are marked into a separate group labeled as Region 0 in Figure 3. The remainder cells are separated based on contiguous regions. In the example shown in Figure 3, there are three such regions, one outside the geometry (Region 2), one inside it (Region 1), and one inside the inner circular component (Region 3).

Surface Wrapping – In this case, the interest is in generating a volume mesh for the volume topologically represented by Region 1 in Figure 3. Therefore, all other regions are deleted and bounding surface of Region 1 – referred to as the interface – is extracted as shows in Figure 4. This surface is made up of squares. Next these squares are split into triangles and the nodes of the triangles are moved onto the closest location on the original geometry. After this step the interface appears as shown in Figure 5. It is now referred to as a “wrap surface” since it covers the original geometry like a shrink wrapper. Notice that the wrap surface walks over problematic areas in the original geometry such as gaps and unnecessary fine details.

However, at this point, the wrap surface is still not capturing some essential sharp geometric features such as sharp corners from the original geometry. Next, to capture these, the wrap surface is modified by performing an “imprinting” operation. In this operation, all the sharp features in the original geometry are identified and the nearby nodes of the wrapper surface are moved to these feature lines. After this, the wrap surface appears as shown in Figure 6.

Post-Wrap Operations – After the initial wrap surface is ready, it is cleaned-up and fine tuned by performing a number of operations such as removing redundant duplicates, correcting self-intersections, inflating thin surfaces, improving skewness of triangles by performing smoothing and swapping operations, and separating the wrap surface, into different parts that correspond with the parts in the original underlying geometry. After these operations, a final wrap surface is obtained which serves as the final surface mesh. This is depicted in Figure 7. The final surface mesh is comprised exclusively of triangular elements.

VOLUME MESH

Once the surface mesh is ready, the second step in the mesh generation process is volume meshing, which fills-

up the region bounded by the surface mesh with volumetric elements. There are several automated techniques for this such as Delauney Method^[4] and the Hex-Core Method^[6]. These methods are described adequately in prior literature, and therefore are not discussed here. In the example discussed here, the Delauney Method has been used to create a volume mesh from the final surface mesh shown in Figure 7. The volume mesh is depicted in Figure 2.

RAPID MESHING FOR VEHICLE AERODYNAMICS

The previous sections discussed the basic principles of rapid meshing with a surface wrapper. Several specifics need to be taken into account when using this technique to create meshes from complex real-life vehicle geometries for vehicle aerodynamics CFD simulations. These are now discussed here.

For conducting vehicle external aerodynamics CFD simulations the mesh needs to be constructed such that it provides the following abilities:

- Ability to provide specific boundary conditions required in aerodynamics studies, such as rotating wheels and moving road/belt, radiators, fans, and others.
- Ability to use specific turbulence models and sub-models such as near wall functions.
- Ability to boost the accuracy of flow solution, by (a) providing the required spatial resolution in all areas of the computational domain (b) limiting numerical diffusion
- Ability to perform specific post-processing needed for (a) extracting aerodynamic forces and moments data and (b) visualization of the flow structure

To provide these abilities, the surface meshes prepared for vehicle aerodynamics simulations need to have the following specific attributes in addition to basic attributes such as water-tightness that have been described earlier in this paper:

- Separate identity for various vehicle parts
- Sharply maintained feature lines on all exterior surfaces. (This attribute is particularly important because surface wrappers tend to break and blunt sharp edges in the geometry. Since the sharpness of feature edges – such as those along the A-Pillar – strongly influences the flow structure in the wake of such features, it is essential to ensure that the wrapped surface mesh captures feature edges sharply.)
- Different resolution in different regions that will allow prominent flow features such as separation regions and sharp flow gradients to

be captured accurately. The resolution distribution should be decided by intelligent defaults and/or be specifiable by the user.

- Gradual change in resolution around any particular point. Sudden changes in surface mesh resolution can not only lead to inaccurate flow solutions, but can also make the prism layer generation process problematic (in the volume mesh generation step).

A commercial implementation of the surface wrapper rapid meshing technology available in the software TGRID5.0^{®[6]} was used in this study. In the current work we have enhanced the surface wrapper and other rapid meshing tools in TGRID5.0 to make them capable of producing surface meshes for vehicle aerodynamics with the above described attributes in a largely automatic way without considerable manual effort. This was accomplished by creating an instruction set for the TGRID5.0 surface wrapper to perform numerous steps that will yield a final surface mesh with the above attributes. We have also added steps for automatically creating a volume mesh from the final surface mesh in this instruction set. As a result, this instruction set which utilizes the surface wrapper and other rapid meshing tools in TGRID5.0 largely automates the CAD-to-Volume-Mesh process for vehicle aerodynamics simulations. The user interaction is reduced to roughly 2 to 4 hours of manual time. Part of this is required for providing basic inputs to the module such as wind tunnel dimensions, user-defined mesh resolution distribution, coordinates of fans and heat exchangers, etc. The remainder of the user's time may be required in some cases for brief troubleshooting and countermeasures. A graphical user interface (GUI) was also created for facilitating and expediting user input and interaction. This instruction set along with the GUI has been programmed into a separate Aerodynamics Module in TGRID5.0.

The Aerodynamics Module mostly operates as a standalone tool and orchestrates the operation of underlying algorithms such as the surface wrapper, prism layer mesh generator, tet or hexcore volume mesh generator, and others. As inputs it takes (a) the original vehicle geometry in faceted STL format and (b) user inputs through the GUI, and outputs the final CFD ready volume mesh.

A significant amount of research effort was spent in the current work in ensuring that the Aerodynamics Module can robustly handle typical real-life vehicle geometries. For this the instruction set was tested on numerous realistic vehicle geometries and was enhanced and fine-tuned based on outcomes of the tests. Rapid meshing for two of these test geometries is presented next as case studies.