

Parametric Shape Optimization

Dhananjay S. Joshi, Xijia Zhu and Kanwerdip Singh

Chrysler LLC

Copyright © 2008 SAE International

ABSTRACT

External aerodynamic simulations are becoming more important because of regulatory pressures on fuel economy improvements and shorter design cycles. Experimental work is typically done on scaled models to get drag and cooling flow information. This is a time consuming process. Numerical simulations might provide a complementary path to get the answers in a timely manner. This paper discusses one such approach.

INTRODUCTION

Drag reduction has become a necessity because of various reasons. Some of the most important ones are described below.

CAFE standards - Recent increased efforts in congress to raise CAFE (Corporate Average Fuel Economy) in congress clearly demonstrate the need to improve fuel economy of all classes of vehicles.

CONSUMER DEMAND - Aerodynamic drag of the vehicle becomes a major factor in fuel economy of the vehicle above 50 mph. All automotive companies are trying to balance the need for vehicles which consumers want to buy (which is mostly affected by styling of the vehicle) and fuel economy to meet CAFE standards as well as for being competitive in the eyes of the consumers.

GLOBAL PLATFORMS - Another reason for automotive companies to look at fuel economy closely is that they are trying to develop vehicles for global markets instead of having regional models. In the last year the fuel prices have gone up significantly (some projections are that a barrel of crude oil may hit the 100 dollar mark in next year or so). Some markets may be more sensitive to fuel prices than others, so one has to develop a vehicle to achieve higher fuel economy.

This dictates that great care must be taken in development of vehicle attributes such as fuel economy in a timely manner. Aerodynamic evaluation of the vehicle has traditionally been done with experimental tools. There is also a need to have a balance between

drag value reduction and meeting the cooling flow requirements. In general these objectives work in an opposite manner. Too much reduction in drag at the cost of cooling flow cannot work in a real scenario. In most of the automotive companies some kind of scaled full vehicle model is tested in a wind tunnel to get information regarding the drag and cooling flow. Typically vehicle development starts with sketching of new themes and then generating a scan image, which in turn is used to create CAD data. After CAD data is available, a clay model is fabricated. This process involves using new body shell geometry along with underbody and underhood data from the new vehicle or older vehicle with similar architecture. In early phases of the program several themes for the vehicle exterior need to be evaluated quickly. Once themes are narrowed down to a few, one needs to look at optimizing the shape further. With advances in CFD technology from mesh generation, and solver and morphing perspectives it is a worthwhile effort to augment the aerodynamic development of the vehicle. Instead of just evaluating the CAD data available, one can now think of giving direction to surface development and lead the design process. By having an idea about which areas of the vehicle surface can be changed (after discussing with all interested groups such as styling, cooling and vehicle development) a series of calculations need to be performed in a quick manner. One such approach has been tried out in our development process which will be explained in this paper.

METHODOLOGY

The methodology described in the following sections explains the CFD technique used and comparison with experimental data generated on a scaled model. Our knowledge of fluid dynamics and experience tells us that flow around a vehicle body will be transient in nature. In the experimental set up, a time average is taken from the transient data collected over sufficient time interval. Based on this assumption a lot of effort has been put into simulating transient flow around the vehicle and then average value is computed based on similar time intervals. This might be required in the final stages of the surface development where major aspects of vehicle proportions and styling elements are somewhat decided. Initially though, the targets for drag and fuel economy are set by the program management or vehicle

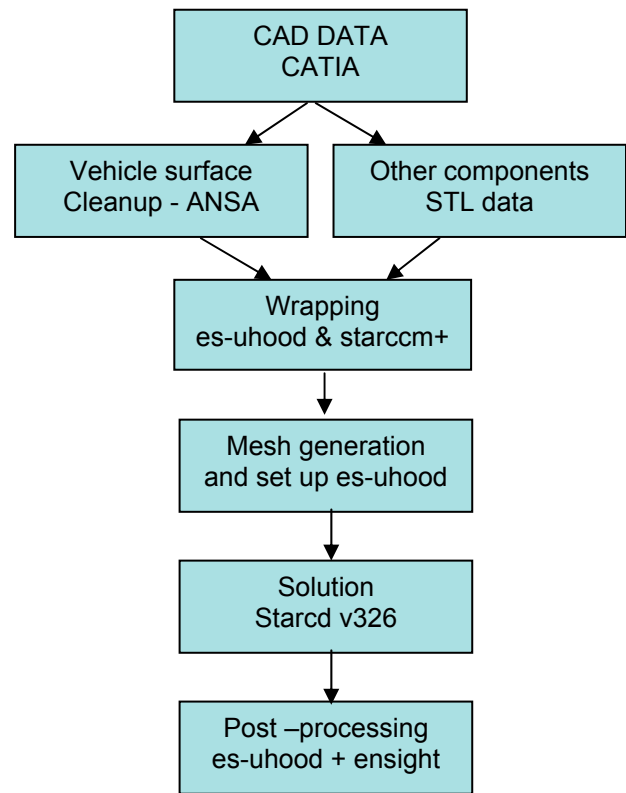
development groups. The information they need are overall values such as drag and cooling flow. To evaluate many themes for vehicle styling in a time efficient way, steady state calculations might provide an economic way. Within one vehicle theme, again, one can perform DOE or similar studies. This requires significant resources when one tries to perform transient simulations. For these reasons, in the current study an attempt is made to provide results for drag of the vehicle based on steady state calculations. Once a baseline drag value is obtained either experimentally or numerically, a desire to improve the drag value is always there. This can be also done numerically in a faster manner.

DATA COLLECTION

In a vehicle development process, designers sketch various themes. Here more importance is given to styling aspects which will appeal to consumers. These sketches are scanned and converted into CAD geometry. After this step, a scaled model is built in a fabrication shop. If the data for underhood and underbody is not available, data from a previous generation vehicle is typically used. As one can see, this process involves lot of man power and time commitment. After getting a model built, wind tunnel testing can proceed. This is also an elaborate process which involves a clay modeler, wind tunnel operator and testing engineers. Once a baseline drag value is established, a series of changes to the surface are evaluated by changing the clay surface. Some changes are feasible but others such as tumblehome change, windshield angle change or a roof header change cannot be accommodated easily. The CFD process used here takes the same data as used for clay model preparation. Then baseline calculation was performed for this study with the same data as used for clay model testing.

Mesh Generation Steps

Traditional mesh generation method and the method used for this project are described in the flow chart below. Cleanup of CAD data is a time consuming process. If the CAD data for the body shell is to be used in conjunction with a previous generation model, a lot of time is typically spent in adjusting data to make a closed surface representation.



Process Flow Chart

Once the data was collected from various sources such as CAD and packaging programs, following steps were taken to build a numerical model.

Wrapping of CAD data

One of the main requirements for a CFD analysis is getting a closed geometry representation. Once that is achieved most of the volume mesh generation tools can be used. With the new approach a wrapping technique was used to reduce time for CAD cleanup. In this case, commercial CFD code STAR-CD was used to wrap whole data. In particular es-uhood mesh generation module was used. This was originally used for underhood analysis but was modified here to suite the requirements for exterior flow analysis. One example of such a wrapped surface is given in figure 1. Engines and transmissions typically involve lot of components and to first collect all CAD data for such an assembly and clean it to get closed triangulation of a good quality requires a lot of time and manual repair work. The triangulation for the wrapped surface is of a good quality and forms a closed envelope.