CAE FRAME WORK FOR AERODYNAMIC DESIGN DEVELOPMENT OF AUTOMOTIVE VEHICLES

[1]Pravin Peddiraju*, [1]Arthur Papadopoulous, [2]Rajneesh Singh
[1]BETA CAE Systems, USA, [2]General Motors Corporation, USA

KEYWORDS – CAE, CFD, Design of Experiments, morphing, shape change

ABSTRACT –

Maximizing fuel efficiency of a vehicle is one of the prime areas of focus in the highly competitive automotive industry which requires development of efficient and optimized vehicle designs. External aero analysis using Computational Fluid Dynamic (CFD) techniques is widely used in the accurate estimation of an automotive vehicle's drag coefficient, often critical in determining the fuel efficiency of the vehicle and thus drives the design development process. In a typical design process, several design variations are analysed, their effect on specific parameters such as the drag and lift coefficients is studied, and thereby, an optimum vehicle design is developed. Numerous techniques can be used to develop design variations from an existing design and to perform design development studies. Morphing is one such technique that offers significant advantages. Using Morphing, design variations can be developed very easily and quickly with minimal effort. In addition, statistical methods can be used to establish correlation between shape change parameters and performance parameters, such as the drag coefficient, to generate optimum design.

In this paper, Meshing and Morphing tools in ANSA, a commercial Computer Aided Engineering (CAE) pre-processing software developed by BETA CAE Systems S.A., are used to develop the baseline Computational Fluid Dynamics (CFD) mesh model and subsequent design variations from the baseline model. Statistical analysis methods are then used to establish a correlation between the geometrical parameters and the drag coefficient to guide the design (shape) development of a General Motors' (GM) automotive vehicle.

TECHNICAL PAPER -

1. INTRODUCTION

In the automotive industry, consumer requirements and competitive environment strongly drive the need to develop efficient vehicles. Efficiency can be improved by developing an optimized vehicle design that meets performance specifications. A typical traditional vehicle development process, as illustrated in Figure 1, includes; 1) Plan, 2) Design, 3) Prototype, 4) Evaluation, and 5) Production steps (1). After Evaluation, the design is further altered and steps 2-4 are iteratively repeated until an optimized design is achieved. With the introduction of CAE into the vehicle development process, the overall cost and time involved in developing an automotive vehicle design has reduced significantly. CAE has extensively simplified the Design phase of the process by replacing expensive physical design studies with computer aided numerical simulations.

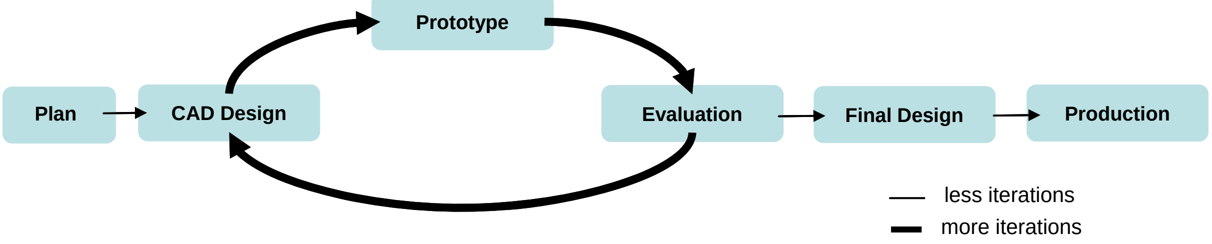

Figure 1 Traditional Vehicle Development Process

The CAE based design process typically starts off with a Computer Aided Design (CAD) model, as illustrated in Figure 2 which is a mathematical representation of the Plan (Concept). Discretized numerical models or meshed models are then built from the CAD using CAE pre-processing software, which are used as an input for numerical simulations of physical testing. These numerical simulations or analyses, which fall under different disciplines, are useful in determining critical performance parameters of a design. Based on the performance parameter values, the design is further fine tuned and the process is repeated till the performance parameters meet objective values. Upon completion of the iterative process, an optimized vehicle design is achieved which is further considered for Prototype, Evaluation, and subsequently Production steps.

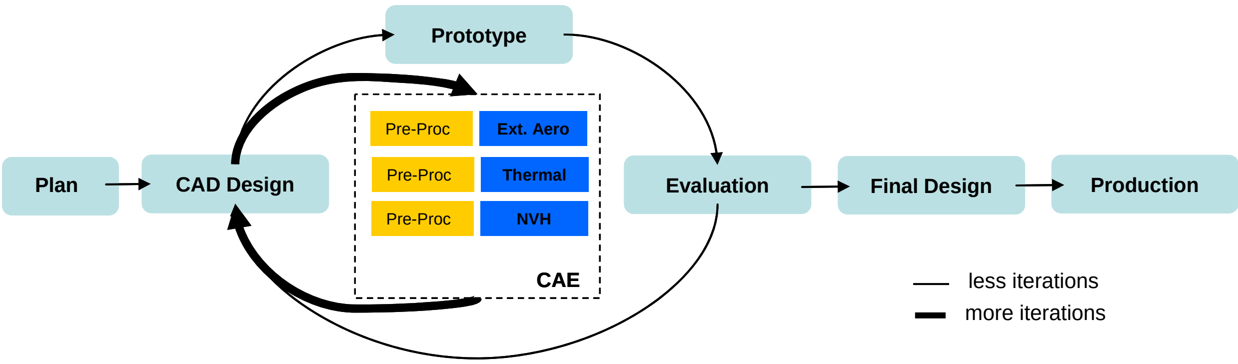

Figure 2 Current Vehicle Development Process

Among the CAE simulations, external aerodynamic analysis is widely used in streamlining a vehicle design to minimize the drag force and is a typical example where analysis assists design. The drag coefficient, which gives a measure of the drag force, can be numerically computed using CFD techniques. The drag coefficient has considerable impact on the fuel efficiency of a vehicle; reduction of the drag coefficient offers an inexpensive solution to improve fuel efficiency compared to improvement in engine efficiency and reduction in overall vehicle weight (2). Figure 3 and Figure 4 illustrate improvement in the gas mileage of an automotive vehicle with reduction in the drag coefficient, which was achieved through aerodynamic design improvement of the car's exterior. The drag coefficient along with the lift coefficient is critical in determining the aerodynamic performance and often plays a crucial role in the aerodynamic shape development of a design.

Photo courtesy of Volvo cars North America

Coefficient of drag, Cd – 0.36

Gas mileage (combined) – 18 mpg†

Figure 3 Volvo 960 (Boxy rear end)

Photo courtesy of Volvo cars North America

Coefficient of drag, Cd – 0.28

Gas mileage (combined) – 23 mpg†

Figure 4 Volvo S80 (Smooth rear end)

The CAE framework often employed in the automotive industry, illustrated in Figure 2, involves iterative improvement of the CAD, based on the performance parameter values obtained through multiple CAE analyses. In most cases, the CAD is modified only after the completion of all CAE simulations. In addition, significant amount of time is spent in developing mesh models for each modified version of the CAD. This procedure of design iterations, although more efficient and less expensive compared to the traditional approach, illustrated in Figure 1, is still time and cost intensive.

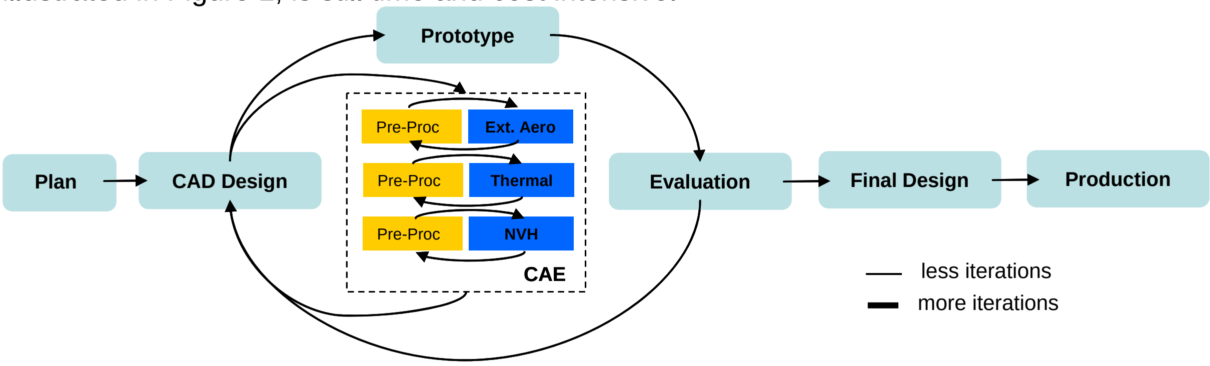

Figure 5 Proposed CAE frame work

A modified CAE frame work is proposed in this paper, as illustrated in Figure 5, which utilizes mesh morphing techniques to develop design variations from a mesh model instead of CAD (4). The proposed frame work also utilizes statistical methods to build correlation between shape change parameters and the performance parameters, which can be used to effectively determine the optimum design configuration. Such a frame work requires evaluation of minimal number of design variations and ultimately reduces the number of iterations between CAD and CAE required in the development of an optimum CAD design. The subsequent sections address the details of the proposed framework with primary focus on the aerodynamic shape development of a GM car.

2. AERODYNAMIC DESIGN DEVELOPMENT

The process employed for aerodynamic shape development can include either a direct optimization or an indirect optimization. In the direct approach (5), an optimization software is typically used which employs optimization algorithms to drive a design to an optimum configuration in the design space taking into consideration the objectives and the constraints. Such an approach often requires large number of numerical simulations. Since CFD simulations use high resolution models for a complete vehicle, where the number of cells can be around 20-30 million, it currently takes significant amount of time to complete the optimization process. In the indirect approach, the design space is sampled at appropriate points and a response function is built to describe the behaviour of the objective function over the design space. This response function is then used to determine the optimum design point. Since the response function is built based on few points, the number of simulations required is thus less compared to the direct approach. As the Aerodynamic simulations for automotive vehicles are quite resource intensive, therefore, the later approach is often employed by CAE engineers for quick design development studies.

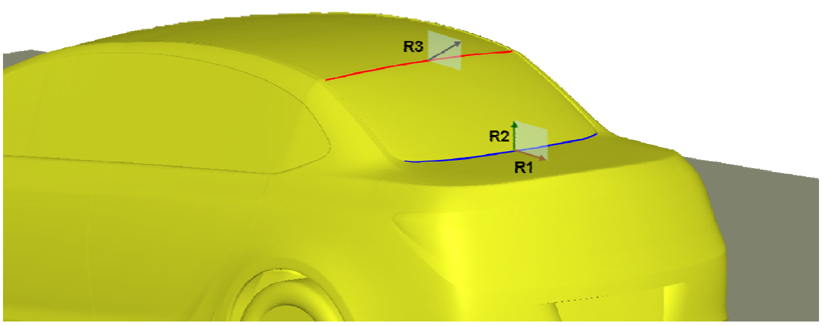

Figure 6 Shape Change Control Factors

In the aerodynamic design development of a car, the shape of its rear end has significant influence on the aerodynamic flow and thus on the drag (6). The critical features that define the shape of the rear end of a car include deck and rear roof edges. The aerodynamic flow can be greatly controlled by modifying the shape of these two features. In the current work, three control factors, illustrated in Figure 6, are used to modify the shape of the above mentioned features. Four design configurations were generated for the control factors using an L4 orthogonal array construction. The details of the four design configurations are enumerated in Table 1.

Control Factor	Level 1	Level 2
Displacement of Deck Edge in X-direction (R1)	-100	100
Displacement of Deck Edge in X-direction (R2)	-10	10
Displacement of Roof header line in Normal direction (R3)	0	50

Table 1 Control Factor values for the studied designs

Quick investigation of design variations requires faster generation of accurate CFD mesh models from CAD, and design variations from the CFD mesh models. The integrated pre-processing environment in ANSA, which includes geometry healing, meshing and morphing modules, provided necessary tools to build the required CFD models. The following sections furnish additional details regarding the pre-processing activities, the CFD simulations and the design variation studies.

Pre-processing

For the current study, ANSA was used to carry out the pre-processing activities which included; 1) CFD grid generation, and 2) Parameterized Morph set-up. As a first step, ANSA Translator was used to translate the native format Unigraphics (UG) CAD data into corresponding ANSA database(s). ANSA Translator uses UG libraries for CAD translation which provides better interpretation of CAD topology as defined in UG. This provides better translation quality for the geometry data compared to the process where UG data is converted to neutral CAD format data, such as IGES or VDA-FS or STEP, and then directly imported into ANSA. Better translation of CAD data ensures a lesser amount of time spent on preparing CAD for CFD meshing. The next sections describe in detail the grid generation and morphing set-up steps.

Grid Generation

The CAD data translated into ANSA typically contained solid description for parts with both inner and outer surfaces. From this data, the wetted surfaces were extracted using ANSA's geometry isolation tools to prepare a condensed CAD model. Then, the surfaces were fixed for any geometry issues, and a water tight geometry model was constructed. The geometry healing and construction tools in ANSA were extensively used in preparing a Common Digital Model (CDM) – the water tight model with cleaned geometry - from the condensed CAD model. The common digital model serves as a basis for analysis specific mesh generation. As ANSA maintains Geometry-Mesh association, usage of CDM offers great flexibility in generating discipline specific high fidelity mesh which is often a requirement for CFD analyses.

The vehicle model was embedded in a virtual wind tunnel, constructed using the geometry creation tools in ANSA. The wind tunnel was extended by 2 car lengths in front of the car, 5 car lengths behind the car and 2 car widths along the sides.

ANSA's Batch Mesh manager was used to generate the mesh, which included surface, boundary layers and volume mesh, for the complete model. The Batch Mesh manager provides a frame work to organize the model into multiple scenarios based on the type of mesh generated, and further into multiple sessions under each scenario depending on the mesh specifications and the quality criteria. Each of the sessions can contain multiple parts, which can be filtered using advanced filtering options. Once defined, the Batch Mesh frame work can be used repetitively for mesh generation for other similar models.

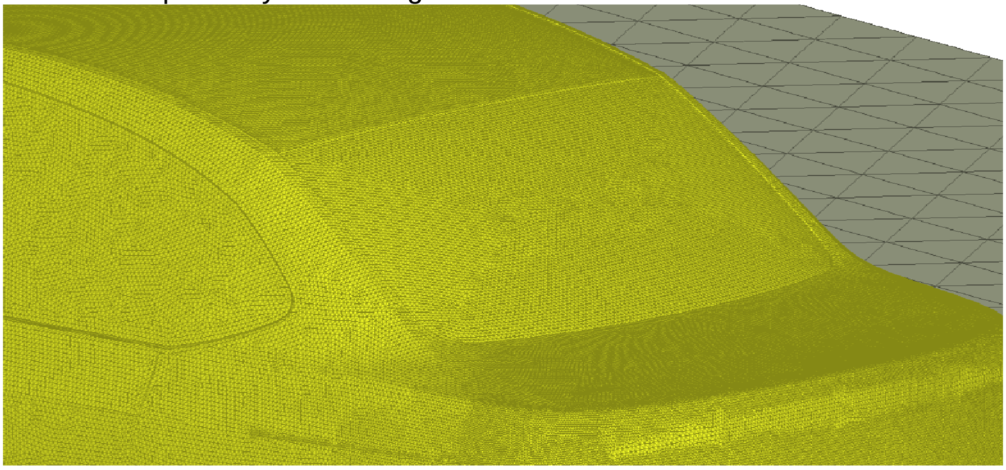

Figure 7 Surface mesh on the vehicle

The surface mesh generated on the model was of a varying size triangular mesh, with fine resolution mesh on the front portion of the car, relatively coarse resolution mesh on the rear portion and with gradual transition in between. Figure 7 illustrates the surface mesh generated on the vehicle and the wind tunnel. Close proximity regions present in the model were identified based on a factor of the local element size using proximity detection tools. Such regions pose mesh quality issues for the volume mesh. The identified regions were automatically fixed for proximity issues by auto refining the local mesh.

Boundary layers were then generated from the styling surfaces that critically affect the air flow over the vehicle. Pentahedral or Prismatic elements were generated to capture the boundary layer effects accurately. A total of 6 layers were generated, with an initial height of 1.0 mm and a growth rate of 1.2. Any problematic regions such as the acute angled regions and the close proximity regions were automatically identified and omitted from layer generation process. Such a feature facilitates generation of boundary layers on non-problematic regions, and leaves sufficient gap in the problematic regions to generate a good quality volume mesh.

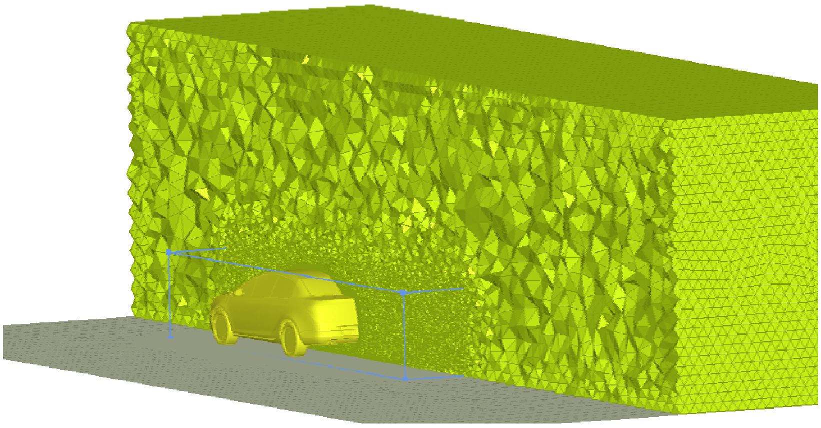

Figure 8 Volume mesh around the car

For the external aero analysis, five volumes:1) Main volume between the vehicle and the virtual wind tunnel box, 2) Heat exchanger, 3) Radiator, 4) Condenser, and 5) Radiator fan were meshed with tetrahedral elements. An orthogonal “size box” circumscribing the vehicle, extending by 0.5m in front of the car and 4m behind the car, was created. The size of the volume mesh was restricted to 60mm in the size box region, as illustrated in Figure 8, to accurately capture the wake. 22million cells were generated in total for the complete model. The skewness of the tetrahedral elements, computed according to FLUENT's (commercial CFD analysis software) equi-volume definition, was kept below a value of 0.97.

Morph Set-up

To create design variations from the baseline model (un-morphed configuration), a domain based morph set-up was created for the rear windshield region of the vehicle, as illustrated in Figure 9. The morph box edges closer to the deck lid, rear wind shield and the C pillar regions were fitted onto the design features for accurate shape changes. Three morphing parameters that correspond to the three control factors illustrated in Figure 6 were defined to precisely change the shape of the critical design features.

An Optimization task was created using ANSA's Task Manager utility to organize the morphing parameters, and to simulate the design variations. Simulation of design variations using the Optimization task helps in evaluating feasible configurations through visualization. Such functionality in turn eliminates the need to evaluate all theoretically possible variations that are not physically feasible to be implemented, and thus facilitates setting of appropriate values for the control factors. A custom ANSA script was developed and incorporated as a check item in the Optimization Task to inspect and fix the mesh quality at the end of each morphing operation. As a last step, the Optimization task exports the model in FLUENT format for CFD simulations.

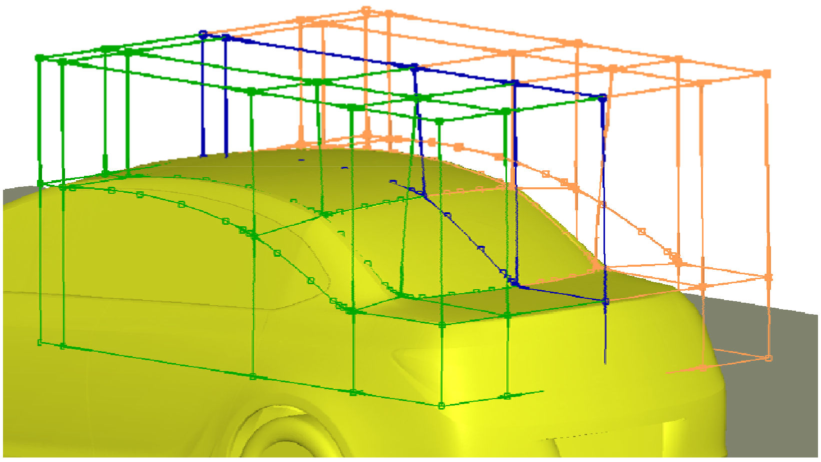

Figure 9 Morph box set-up

<u>External Aero Analysis</u>

FLUENT, a commercial CFD solver was used to perform the external aero analysis to estimate the drag and lift coefficients of the baseline model and its design variations. The completed CFD model was exported from ANSA and then read into FLUENT. An automated procedure developed at GM was used to set-up the boundary and operating conditions and other solver inputs. The automated procedure ensured robustness and consistency for the simulation process employed for each design.

A no-slip boundary condition was used on all the vehicle surfaces. Symmetry boundary conditions were imposed on the tunnel surfaces. Analysis was performed at 110 kph vehicle speed condition. Second order discretization schemes were used for improved accuracy. The Realizable k-e turbulence model with non-equilibrium near wall function was used. The lift and drag coefficients, C_D and C_L, were then estimated from the analysis results for each design. Each simulation required about 500 CPU hours on an IBM high performance computer.

Design Variation Studies

For the current study, four design variations were investigated in addition to the baseline model. The design variations were generated based on the control factor values generated using orthogonal Design of Experiments (DOE) set-up. The physical feasibility of these design variations was verified using the simulation tool available in ANSA's Optimization task. Figure 10 illustrates the cross-section of the vehicle along its symmetry plane for the baseline design and its four design variations.

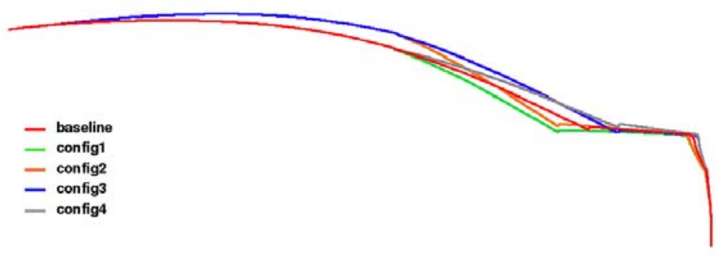

Figure 10 Schematic of vehicle cross-section for the studied designs

	R1 (mm)	R2 (mm)	R3 (mm)	ΔC_D	ΔC_L
Config 1	-100	-10	0	0.000	0.002
Config 2	-100	10	50	0.011	0.025
Config 3	100	-10	50	0.007	0.016
Config 4	100	10	0	0.005	0.011

Table 2 Incremental drag and lift coefficient values for the studied designs

Incremental drag and lift coefficients were computed for the four design variations with respect to the baseline design. Table 2 enumerates the control factor values for the four design variations along with the incremental drag and lift coefficients. This data is used to build a correlation (response) function between the vehicle performance parameters and the geometric parameters using statistical analysis tools. The statistical analysis requires evaluation of performance parameters at minimal number of design configurations and performance parameters at any other intermediate configurations can be estimated using the response function.

Figure 11 Main effects plot for incremental drag coefficientsillustrates the main effects plot for the estimated incremental drag coefficients. The pictures illustrate variation of incremental drag coefficient, ΔC_D with the average control factor values. It can be easily interpreted from the plots that the control factor R_3 seems to have maximum impact on the drag coefficient, in contrast to control factor R_1, which has the least impact. Such plots provide valuable feed back in understanding the design features that critically influence the aerodynamic flow and thus the drag coefficient.

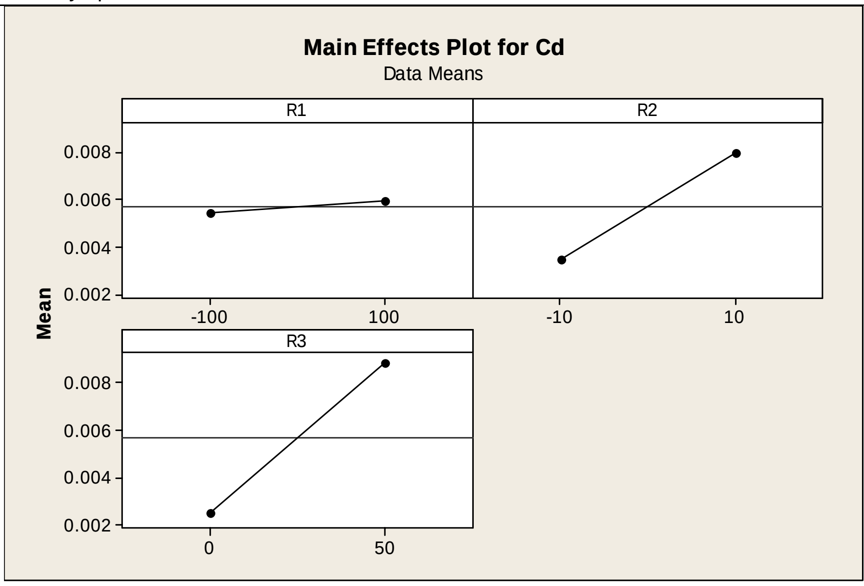

Figure 11 Main effects plot for incremental drag coefficients

In addition to the main effects plot, a surface plot was generated as illustrated in Figure 12 to depict the variation of drag coefficient with critical geometrical parameters, in this case, R_3 and R_2. Only a planar response could be generated as the design was sampled at two levels. With more sample points, any non-linearity in the drag coefficient behaviour can be effectively captured. From the plot it can be concluded that the drag coefficient increases with increase in the values of R_3 and R_2. Thus, the drag coefficient can be minimized by lowering the values for R_2 and R_3.

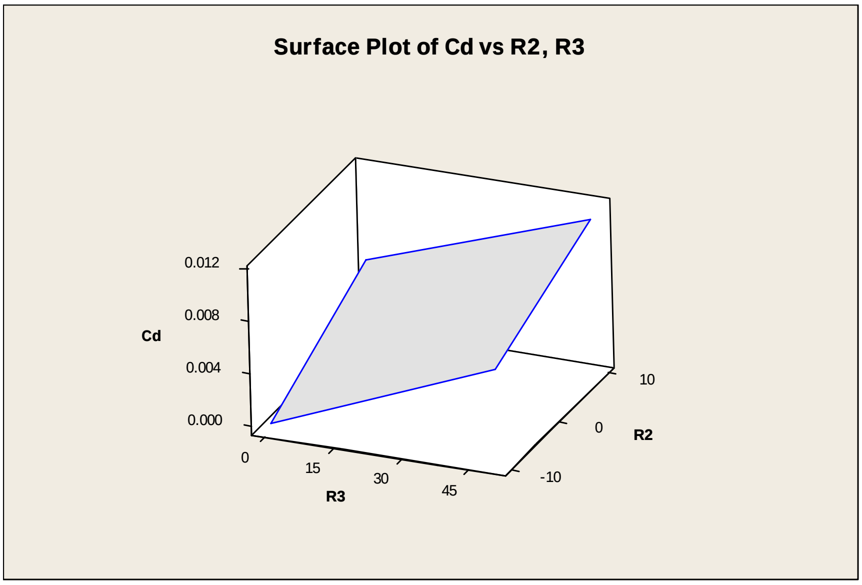

Figure 12 Surface plot of drag coefficient

3. SUMMARY

Developing an efficient and optimized vehicle design is one of the prime objectives of the highly competitive automotive industry. One aspect of developing an efficient vehicle design is to improve the fuel efficiency of a vehicle by minimizing the drag coefficient, often predicted using CFD techniques such as the external aero analysis. This involves intricate Design development studies that ascertain the affect of aerodynamic shape on the drag coefficient.

A detailed CAE frame work is presented in this paper which has been employed to perform Design development studies on a GM automotive vehicle for better aerodynamic performance. The meshing and morphing capabilities of ANSA are demonstrated by performing a 3 factor parametric shape development study of the rear end of a passenger vehicle. Designs for an L4 orthogonal array were created by morphing the baseline mesh model. The ANSA Batch Mesh manager facilitated generation of Surface Mesh, Boundary Layers and Volume Mesh in an automatic manner, saving considerable amount of modeling time. The Morphing tool and the Optimization Task utility provided necessary tools to define and simulate complex shape changes. The Optimization Task can also used to generate necessary design variations in batch mode, eliminating the need for user interaction, which can facilitate easy integration of ANSA with external optimization software for DOE and Optimization studies. During significant shape changes, the mesh can often deform significantly affecting its quality. For such cases, the integrated environment in ANSA provides all necessary tools to fix the mesh quality or regenerate the mesh locally. ANSA's meshing and morphing tools can be effectively used to develop CFD models and their design variations with minimal effort. The baseline model and the design variations were subsequently analyzed for aerodynamic performance. Aerodynamic drag and lift coefficients were computed to create a database for statistical post processing.

The CAE frame work discussed in this paper incorporates the usage of a pre-processor with an integrated environment of meshing and morphing to make necessary shape changes and statistical analysis to guide the design development process. Such a frame work minimizes the number of iterations required to develop an optimum design and consequently, reduces the design development time and cost.

REFERENCES

(1) KOJIMA, Y – Mechanical CAE in Automotive Design, R&D Review of Toyota CRDL, Vol. 35, No. 4

(2) http://www.edmunds.com/advice/fueleconomy/articles/106954/article.html

(3) GENERAL MOTORS – 2005/2006 Corporate Responsibility Report, http://origin.gm.prod.gmgssm.com/corporate/responsibility/reports/06/400_products/1_ten/410.html

(4) CAMPOS, F., GEREMIA, P., SKAPERDAS, E.- Automatic Optimization of Automotive Designs, Proceedings of 2nd ANSA & META International Congress, 2007, Thessaloniki, Greece

(5) SINGH, R - Automotive Aerodynamic Design Optimization Process for Automotive Vehicles, SAE Congress, 2003, Detroit, USA

(6) HUCHO, W - Aerodynamics of Road Vehicles, SAE International, 1998, Detroit, USA

(7) BETA CAE SYSTEMS S.A. - ANSA v13.0.1 Users Guide