CFD-A Trend in Automobile Aerodynamics Technology

RAHUL RAMESH JOSHI[1], AMIT APPASAHEB PATIL [2]

[1,2] Assistant professor, Dept of Textile Engineering, TEI Ichalkaranji ,Maharashtra,India

---

Abstract - *Aerodynamics in automobile plays a crucial role, how will they behave on road & how efficient they performs the task they are designed for. For all vehicles ranging from small passenger vehicles to commercial busses & trucks, reducing air drag is one of the most efficient ways of improving fuel economy. Optimum aerodynamic design has to produce the best balance between drag that allows the car to be stable & drivable at high speed. In this paper we study the optimum aerodynamics design & basic steps to be followed during the design using Computational Fluid Dynamics.*

***Key Words*: Aerodynamics, prototyping, best balance, economy.**

1. What Is CFD?

Computational fluid dynamics (CFD) is the computational science of simulating fluid flows by using computers. CFD is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.) CFD made possible by the advent of digital computer and advancing with improvements of computer resources. CFD provides numerical approximation to the equations that govern fluid motion. Application of the CFD to analyze a fluid problem. First, the mathematical equations describing the fluid flow are written. These are usually a set of partial differential equations. These equations are then discretized to produce a numerical analogue of the equations. The domain is then divided into small grids or elements. Finally, the initial conditions and the boundary conditions of the specific problem are used to solve these equations. The solution method can be direct or iterative. In addition, certain control parameters are used to control the convergence, stability, and accuracy of the method.

1.1 Why to Use CFD?

Basically, the compelling reasons to use CFD are these three:

1. Insight

There are many devices and systems that are very difficult to prototype. Often, CFD analysis shows you parts of the system or phenomena happening within the system that would not otherwise be visible through any other means. CFD gives you a means of visualizing and enhanced understanding of your designs.

2. Foresight

Because CFD is a tool for predicting what will happen under a given set of circumstances, it can answer many 'what if?' questions very quickly. You give it variables. It gives you outcomes. In a short time, you can predict how your design will perform, and test many variations until you arrive at an optimal result. All of this is done before physical prototyping and testing. The foresight you gain from CFD helps you to design better and faster.

3. Efficiency

Better and faster design or analysis leads to shorter design cycles. Time and money are saved. Products get to market faster. Equipment improvements are built and installed with minimal downtime. CFD is a tool for compressing the design and development cycle.

1.2 A Case Study On Opel Astra:

Optimum Design Procedure:

CFD flow simulation can thus be used to readily make statements about flow circulation around a car to the point where models or prototypes are not available.

Offering comprehensive information to designers about the entire flow field can therefore accelerate the process of aerodynamic design. The flow field around a vehicle is physically very complex.

The efficiency of an aerodynamic CFD simulation depends on the time span required to achieve the first set of results, as well as the accuracy of the simulated flow quantities. The turn around time and confidence level in the predictions is two major criteria for success that compete with one another. Creation of the model geometry, discretization of the physical domain, and choice of a suitable numerical computing scheme are significant factors that can determine the level of success of such an effort.

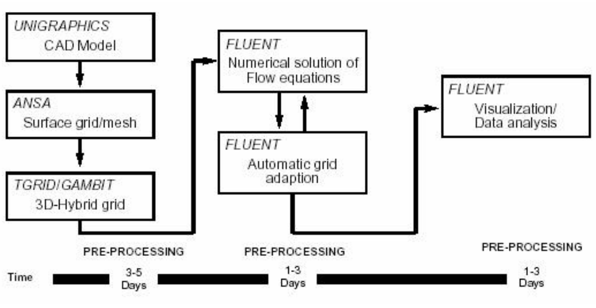

Fig.1. The Design Process

The underlying simulation process is divided into the following steps: CAD surface preparation, mesh generation, CFD solution of the fluid flow, mesh adaption, and visualization of the results. The software packages used for these steps include, UNIGRAPHICS (CAD), ANSA (CAD/mesh generation), TGRID and GAMBIT (additional mesh generation), and FLUENT (solver and post processing). An elapsed processing time of between 5 and 11 days, without CAD modeling, is usually required for the base case simulation of a complete model, comprised of about 3 million cells. This time scale depends on the complexity of the model. Once the base case has been built, individual modifications can be realized in a fraction of this time span (Fig.1).

2. CAD-MODEL:

To begin the process, the CAD body shell data for the ASTRA is downloaded from a common CAE database at OPEL, from which all of the vehicle parts and components can be accessed. For initial concept studies, the vehicle exterior data is made available by the design department. Underbody data can also be downloaded from the database. For early theme studies there is usually a simplified generic underbody model available, which has already been integrated and meshed. For subsequent aerodynamic studies, individual engine parts are not modeled. In this study, the air cooling vents and the engine space above the sub frame are closed.

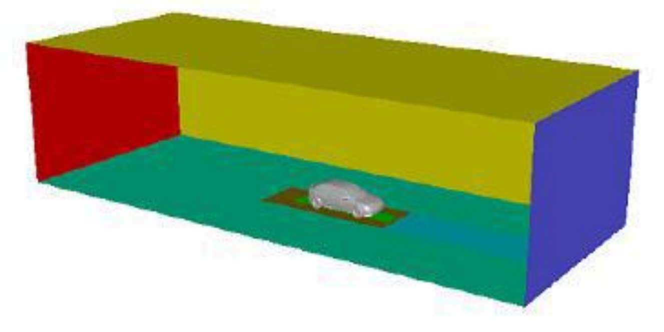

Fig.2. CAD Model

Using this finished base case, numerous types of vehicles can be generated on the same template for simulation in a short time period. A half model with a centerline symmetry plane is sufficient for initial design studies, but the ASTRA in this study is constructed as a full model with an asymmetric geometry (where the asymmetries are mostly confined to the underbody). This is done to enable detailed predictions of the flow field around and under the vehicle. The wind tunnel geometry around the model is a rectangular enclosure. It is of such a dimension that the adverse pressure effects between the vehicle and the wall are minimized. The three CAD components, namely the outer body of the vehicle, the underbody with tires, and the wind tunnel together constitutes the CFD model

2.1 Surface Grid:

The integration, preparation and clean up of the CAD components described above was done with the program ANSA .To do this, the CAD data was exported in IGES format and read into ANSA, where production of a closed topology of CAD areas and definition of macro areas was performed. Additionally, auxiliary surfaces were generated, so that separately meshed fluid zones could later be linked or delinked from one another. These areas and surfaces were then used to generate the surface grid. The following description of the task of grid creation is limited to the styling surfaces, since the wind tunnel and the underbody have already been meshed. The vehicle body is meshed with triangles having a side length of about 20 mm, in as uniform a manner A possible. A minimum side length of about 5 mm used in the areas of the side mirror and the A-pillar. The window frames are covered with regular quadrilateral elements so as to close the stage of the window later using separately inserted prism blocks. Experience shows that generation of a fine and uniform grid on the vehicle body is necessary to obtain realistic values for drag and lift coefficients. Fig shows the shaded surface mesh of the ASTRA

2.2 Generating The 3d Hybrid Mesh:

A hybrid mesh comprised of tetrahedral and prismatic elements was chosen for the CFD model of the ASTRA. This was done so as to ensure that an extensive automatic grid generation of the complex geometry could occur. The volumetric grid was built in TGRID, using the surface grid that was generated by ANSA. It was then exported in a FLUENT format. On the one hand, this means that a high quality resolution of the surface mesh is necessary to create a good volume grid. On the other hand, any modifications must only be performed in the two-dimensional “space” that is the surface mesh. This therefore simplifies and accelerates the task of grid generation by taking into account the possibility of further simulations of various base vehicle variants. To improve the resolution of the oncoming flow boundary layer, the styling surfaces of the vehicle and the floor of the wind tunnel were made up of multiple prism layers. In this instance, avoiding three-dimensional corners or steps in the surface to form a grid with prisms was effective. This help’s generate cells of good quality. The set-in side windows of the ASTRA model along with their corresponding rectangular windowsills are not well suited for direct coverage by prism layers, because distorted prismatic elements would be created. The same is true for geometric steps in the areas of the cowl and A-pillar, and also for inserts and steps in the front and rear of the vehicle. In ANSA, these areas are closed in advance, using the auxiliary surfaces mentioned above. The limiting areas are filled with either tetrahedral or, as in the case of the side windows, regular hexahedral blocks generated in GAMBIT.

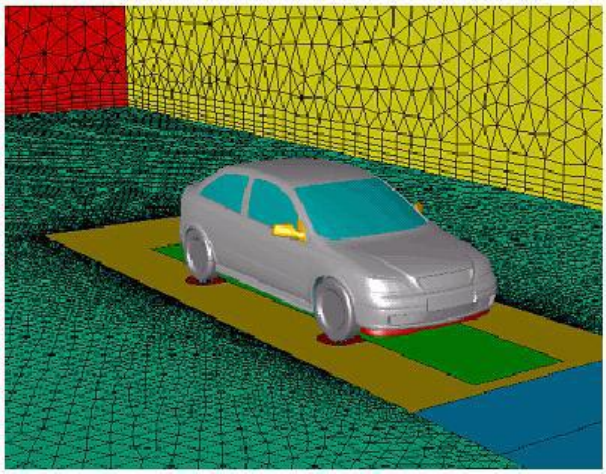

Fig.3. 3D-Hybrid Mesh

3. CONCLUSION

Computational Fluid Dynamics (CFD) gives you a means of visualization and understanding your design as it is a tool for predicting what will happen under the given set of circumstances before physical prototyping. CFD helps you to design better and faster, it provides numerical approximations to the equations that govern the fluid motions. In CFD, governing equations are solved with the help of computer software.

REFERENCES

www.flowfluent.com/solutions/automobile/aero.html

Tao Xing & Fred Stern," Introduction to CFD" IIHR-hydro science & Engg., University of Iowa,2002.

Ing. Andreas Kleber " JA123-Simulation Of Air Flow Around Opel Astra Vehicle With Fluent",2001

GRIEBEL M.,DORNSEIFER T. AND NEUNHOEFFER T."Numerical Simulation in Fluid Dynamics: A Practical Introduction". SIAM Monographs on Mathematical Modeling and Computation. SIAM 1998.