

Simulation of Air Flow Around an OPEL ASTRA Vehicle with FLUENT

Dipl. -Ing. Andreas Kleber
International Technical Development Center Adam Opel AG

Using the OPEL ASTRA as an example, an efficient process for the simulation of the flow field around the exterior of a vehicle is described. The individual steps involved in the simulation process, with a special emphasis on mesh generation and adaption, are reviewed. In conclusion, various areas of application for aerodynamic simulations in the automotive development process are discussed.

An increasing number of high performance computers and commercial codes for grid generation have become available over the last few years so that we can now calculate the air flow around automobile models in a cost-effective manner. 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. The process of aerodynamic design can therefore be

accelerated by offering comprehensive information to designers about the entire flow field.

The flow field around a vehicle is physically very complex. The efficiency of an aerodynamic CFD simulation depends, therefore, on the timespan required to achieve the first set of results, as well as the accuracy of the simulated flow quantities. The turnaround time and confidence level in the predictions are 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.

The simulation process for the OPEL ASTRA that is described in this article is based on practical

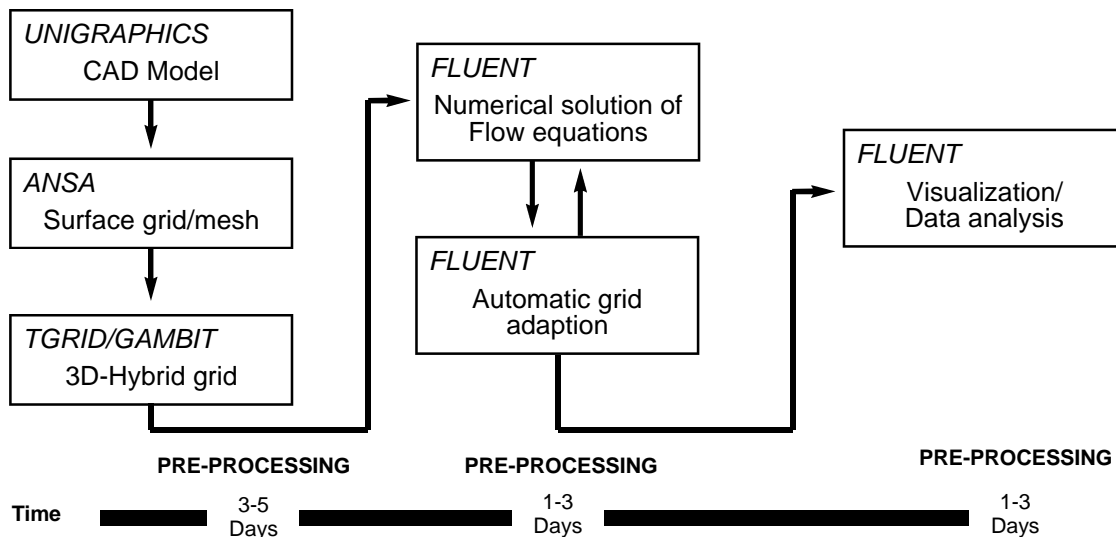


Figure 1: The simulation process

experience, primarily in the field of mesh generation, that has been developed through the use of software from Fluent Inc. from Lebanon, NH, for various automobile projects.

The Simulation 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 timespan (Figure 1).

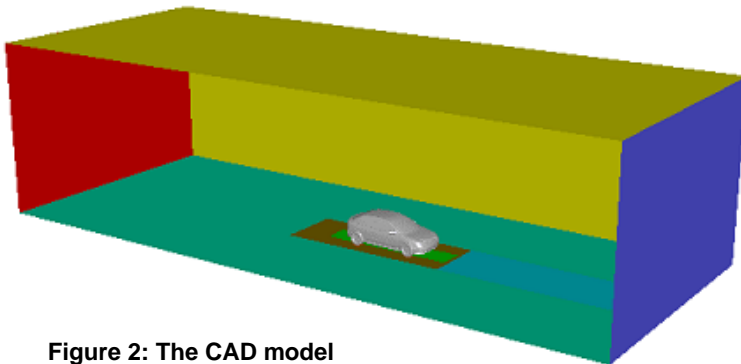


Figure 2: The CAD model

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 subframe are closed. 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 constitute the CFD model (Figure 2).

Surface Grid

The integration, preparation and clean-up of the CAD components described above was done with the program ANSA [1]. 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 as possible. A minimum side length of about 5 mm is 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. Figure 3 shows the shaded surface mesh of the ASTRA.

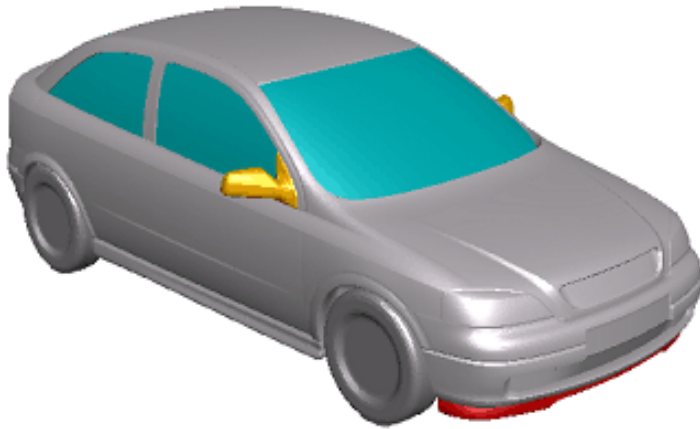
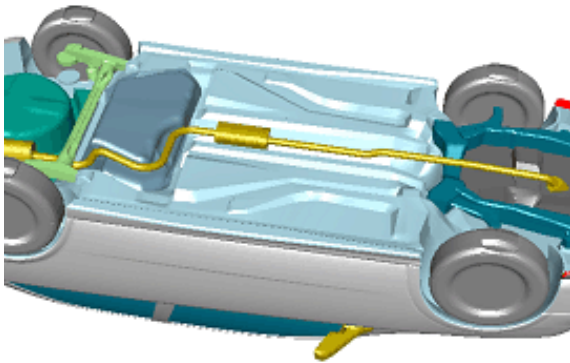


Figure 3: Shaded surface mesh of the CFD model for the vehicle outer skin (above) and underbody (below)



multiple prism layers. In this instance, avoiding three-dimensional corners or steps in the surface to form a grid with prisms was effective. This helps generate cells of good quality. The set-in side windows of the ASTRA model along with their corresponding rectangular window sills 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 tetrahedra or, as in the case of the side windows, regular hexahedral blocks generated in GAMBIT (Figure 4).

Wherever such auxiliary surfaces could not be used to separate areas of complex geometry from the surrounding prism layers, the so-called non-conformal interfaces option was used. In the case of the ASTRA, the external mirrors, the entire underbody, and the wheels are covered using tetrahedra. This means that the prism layers which were generated on the styling surfaces and on the ground, end at the body sill and at predefined areas around the external mirror and wheels. A so-called prism side is generated at the edge of the prism layers. This is a circumscribing zone, which displays quadrilateral surfaces. To couple the quadrilateral surfaces with the tetrahedral mesh of the wind tunnel that is yet to be generated, these are copied and triangulated automatically in TGRID, in correspondence to the distribution of the edge nodes. The triangles that are so created can be manually modified if necessary. Both areas are later handled as internal interfaces. This allows the solver to

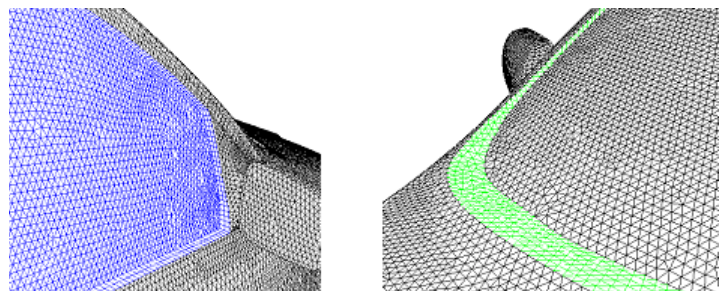


Figure 4: Inserted prism elements (left) and auxiliary surfaces (right)

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 [2]. 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

interpolate the flow data between the different types of elements (Figure 5).

TGRID was used to generate prism layers on the vehicle body shell and ground plane. The height of each prism element is controlled by entering the aspect ratio of the first cell and a geometric growth

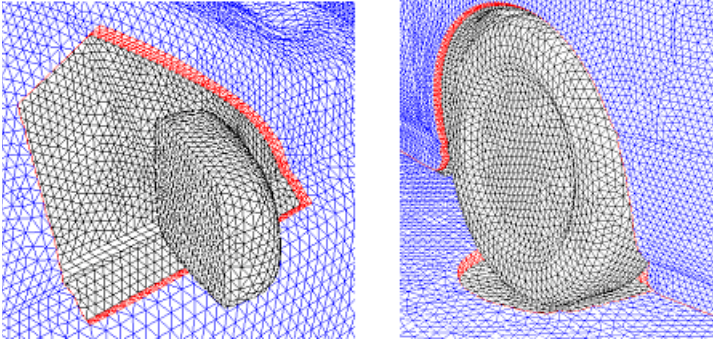


Figure 5: Non-conformal interfaces are shown in red

rate. To do this, meshing parameters were chosen to target a good initial y^+ distribution and a smooth transition to the tetrahedral grid in the external fluid region. The planar side walls of the wind tunnel were automatically split into zones with quad or triangular surface elements, depending upon where the prism layers join (Figure 6). There they were retriangulated. The regions that had not been made into a grid using prisms were grouped together to form their own domain. The remaining volumetric grid generation was completed using TGRID, resulting in a hybrid mesh of approximately 3 million cells. The maximum skewness was less than 0.8 [4].

Numerical Solution of Flow Equations and Grid Adaption

A 3D steady-state, incompressible solution of the Navier-Stokes equations was performed using FLUENT 5 [3]. Turbulence modeling was done with the realizable $k-\epsilon$ model using non-equilibrium wall functions. The free stream velocity was set to be 140 km/h. For about the first 300 iterations, a first order upwind discretization scheme was used to accelerate the convergence.

Thereafter, a second order upwind scheme was used. For simulations of this type, a total of 5 combined grid adaptations are usually carried out during the solution process, to satisfy the y^+ criteria. Statistical pressure gradient adaptation is also executed, acting on about 1 to 2% of the total cell count. Hanging node adaptation (Figure 7) is typically used, which allows for subsequent coarsening of the grid if the need arises later in the solution process.

After each adaptation, about 150 iterations are usually required to converge the solution to the previous residual values. For scaled residuals, the convergence criterion is satisfied if the residuals fall below a value of 10^{-3} . A more accurate measure is when fluctuations in the lift and drag coefficients lie within the range of ± 0.001 . In this particular case, a total of 1600 iterations on an eight processor SGI Origin 2000 were performed (Figure 8).

Post-Processing

In a normal wind tunnel test, flow visualization is either too restrictive or too expensive. In contrast, the CFD simulation process readily generates data for the entire flow field. This can be combined and presented in the manner the user requires. Flow field visualization (surface pressure distribution, for

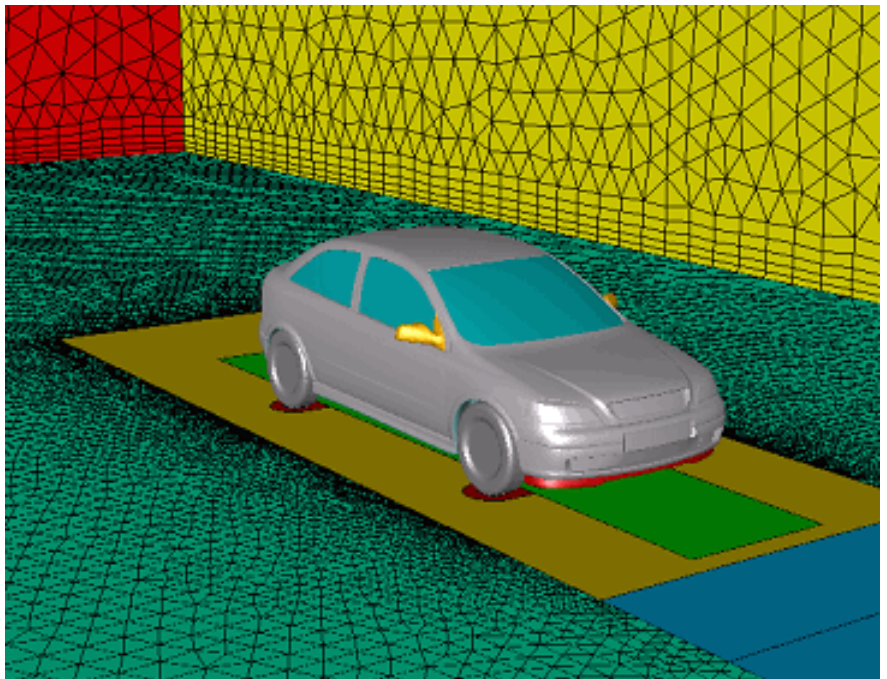


Figure 6: The 3D-hybrid grid

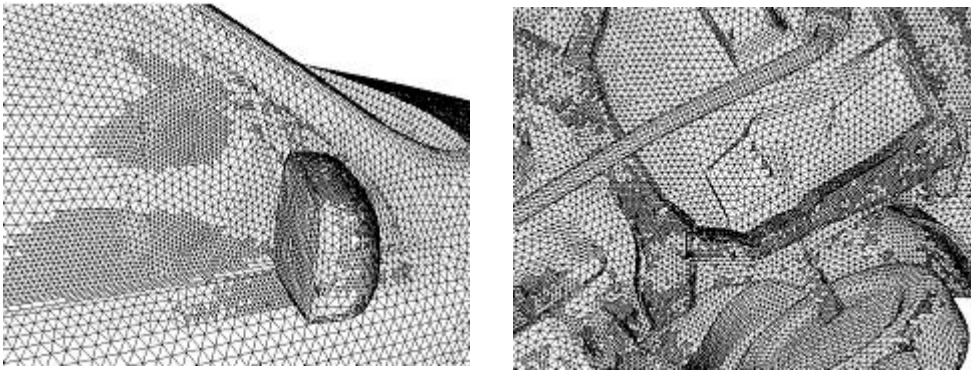


Figure 7: Adapted cells on the vehicle skin

example) assisted by path lines can provide the aerodynamic development engineer with significant insight into the basic flow features. It also indicates areas for further optimization. In addition, general statements may be made about the aero-acoustics and contamination of the vehicle. It can be used to

the wind tunnel test. The drag coefficient usually deviates by 5-10% from experiment. The calculation of lift coefficient is still a challenge. This can be traced back to fundamental limitations with the $k-\epsilon$ turbulence model. In spite of these limitations, the flow simulation process can be used for aerodynamic evaluation of

early design studies based on the computed coefficients. Drag increments (ΔC_D) of two design studies (which differ significantly in shape) can often be predicted exactly using simulation (Figure 10).

CFD flow simulation is a useful tool for providing predictions of pressure distribution and forces exerted on the vehicle components. The experimental costs for measuring these quantities are very high. For example, Figure 11 illustrates how a CFD simulation can be used to investigate the forces on the ASTRA-OPC rear wing and trunk lid.

As a standard, engineers at OPEL compute the aerodynamic forces on the door frames and the side panel of the front door for diagonal flow. The data generated is fed as boundary conditions to the subsequent FEM structural calculation to simulate the bending of the door.

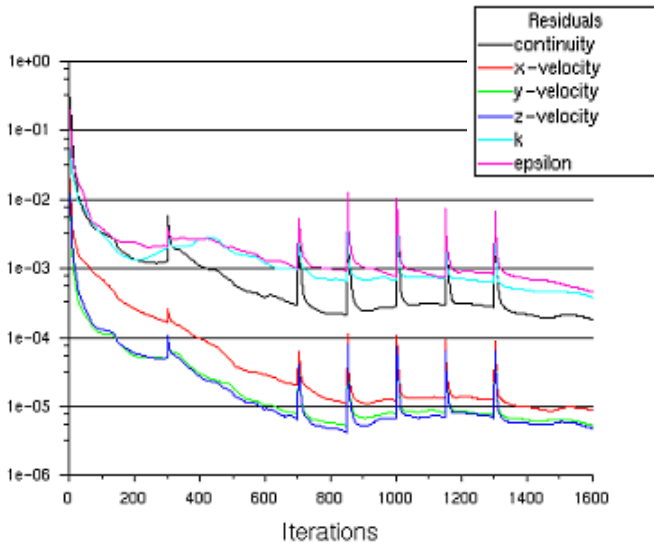


Figure 8: Scaled residuals showing peaks that follow grid adaptations

analyze turbulence energy distribution or the progression of wall surface streamlines in the A-Pillar region and at the side glass. Figure 9 shows examples of flow field visualization.

Evaluation of lift and drag coefficients is critical to vehicle aerodynamics. In this area, flow simulation may not be as accurate as

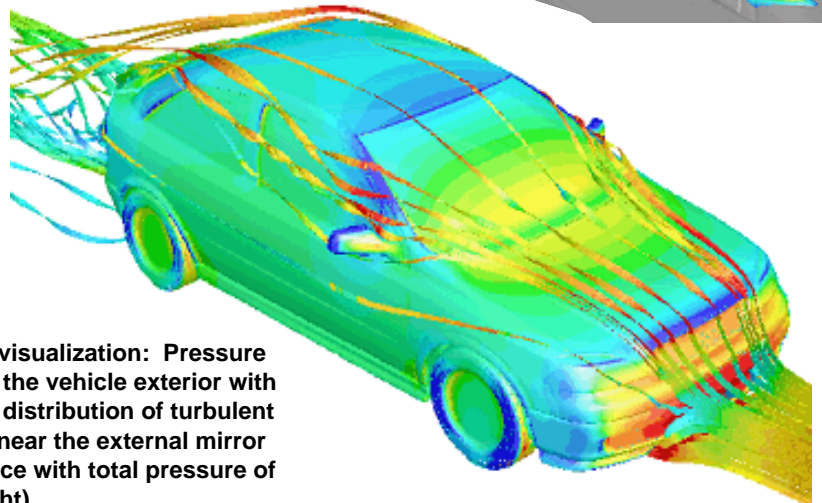


Figure 9: Flow visualization: Pressure distribution on the vehicle exterior with path lines, and distribution of turbulent kinetic energy near the external mirror on an iso-surface with total pressure of zero (above right)

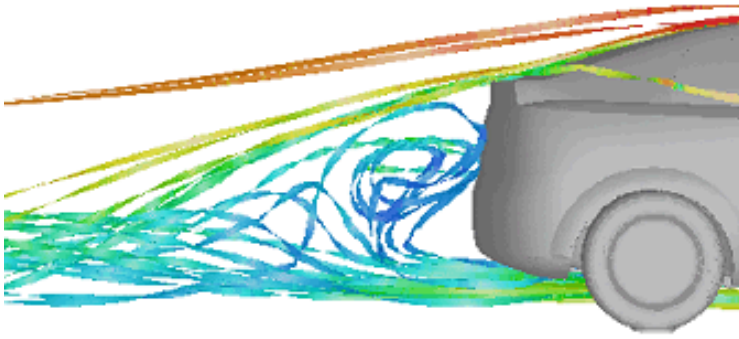


Figure 10: Flow analysis after a design study (path lines)

power available, the complexity of computational models is bound to increase. The use of a comprehensive CFD model with advancement in passenger space and engine flow through is already a possible scenario. The use of high precision computers coupled with advanced applications for the simulation of turbulent flow (e.g. RSM, LES) will contribute to accurate prediction of the flow field around the vehicle [6, 7]. In addition, the effects of of aeroacoustics and water management will play a more significant role in aerodynamic CFD simulations of the future.

Summary

A combination of ANSA and FLUENT was used to develop an efficient process to simulate the flow field around a vehicle. The example of the OPEL ASTRA indicates that the proposed gridding strategy can also handle complex geometry.

On the one hand, flow simulation serves to supplement the experimental work done in the wind tunnel using flow field visualization. On the other hand, aerodynamic coefficients are available for initial design studies, long before hardware models and prototypes come into existence. The CFD computation of aerodynamic forces effectively supports the FEM structural calculations. In view of the exponential increase in computing

References

- [1] ANSA Tutorial, BETA-CAE Systems 1998.
- [2] TGRID 3 User's Guide, Fluent Inc. 1997.
- [3] FLUENT 5 User's Guide Volume 1-4, Fluent Inc. 1998.
- [4] Matus, R., "Modeling External Aerodynamics", Fluent Inc. , User's Group Meeting 1994, Burlington VT.
- [5] Spragle, G. S. & Smith, W. A, "Hanging Node Adaption on Unstructured Grids", Fluent Inc., Lebanon NH; AIAA 1994.
- [6] Francis T. Makowski and Sung Eun Kim, "Advances in External-Aero Simulation of Ground Vehicles Using the Steady RANS Equations", SAE Technical Paper 2000-01-0484.
- [7] Piomelli, U. Chasnov, J. R., "Large Eddy Simulations: Theory and Applications", ERCOFTAC Summer School on Turbulence Modeling, Stockholm 1995.

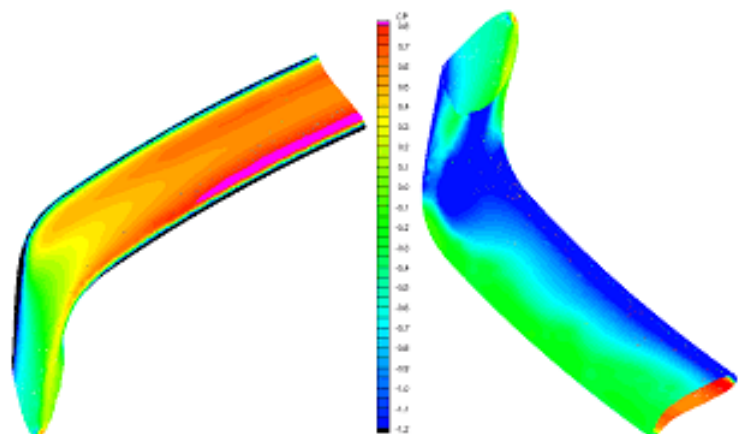


Figure 11: ASTRA OPC with rear wing (above) and pressure distribution on the rear wing (right)