CFD Meshing for Aerodynamics of the Volvo XC 90

With the aim to shorten the lead times for the aerodynamics simulations with CFD programs of Ansys Fluent significantly, the Volvo Car Corporation has analyzed, optimized and partially automated all process steps and tools. One of the changes was the use of the meshing program Harpoon from CEI as an alternative to the programs ICEM-CFD and TGrid, uses so far. To show that these changes were not made at the expense of quality, a comparison was made between simulations for a Volvo XC 90 on the base of an ICEM-CFD/TGrid mesh and a Harpoon mesh.

1 Introduction

In recent years the pressure on the automotive industry has increased extremely. The necessity to develop more and better products in a shorter time span has led to an increased deployment of numerical methods like the CFD simulation. Computational Fluid Dynamics (CFD) not only improves the knowledge and understanding of flow phenomena, but it is a crucial tool to minimize the use of prototypes and test cars. Since more than a decade CFD is used for the vehicle development at Volvo cars in different phases. The broad spectrum reaches from aerodynamics and aero acoustics to under hood flow for cooling performance and climate control. In the early nineties, up to four months were needed for the analysis of quite simple models. Increased computing power and more efficient meshing techniques have reduced the time for a detailed aerodynamic simulation to four up to five weeks (2004). Amongst other things, a further reduction was prevented by a much higher geometrical complexity.

To decrease the project times and the change cycles, the complete simulation cycle was analyzed in detail and improved where ever possible during the last two years. Especially the time consuming steps for the geometrical conversion of the meshing were in the centre of interest, **Figure 1**.

2 Building the CFD Model

Traditionally, at Volvo Cars two meshing programs were used. ICEM-CFD for an Octree mesh for the upper body, and TGrid for an unstructured Delaunay mesh for the under body. Today, meshing is done consequently with Harpoon from CEI, an octree mesher, which generates a hex-dominant mesh. To evaluate the influence of the two different meshes on the simulation results, two variants of the SUV Volvo XC 90 (with and without under body panels (UBP), **Figure 2**) were calculated. It is an exterior model with rear view mirrors, fully detailed under body, fairly detailed engine compartment and open rim wheels.

2.1 Find Vehicle and Extract CAD Data

For the management of the CAD data Volvo Cars uses the PDM solution Team-Center from UGS. Users now can find and extract easily the CAD models of a specific car (for example left hand drive, diesel etc.). The CAD data from TCe is saved in the JT-format, which is universal and contains both CAD surfaces (Nurbs)

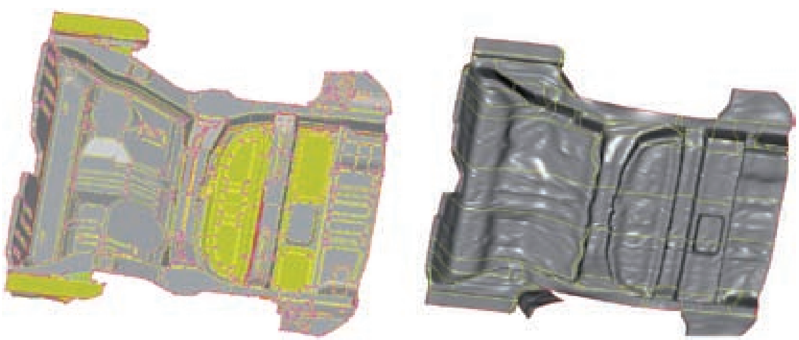

Figure 1: Comparing original CAD part (left) to Catia V5 reconstructed CAD geometry (right)

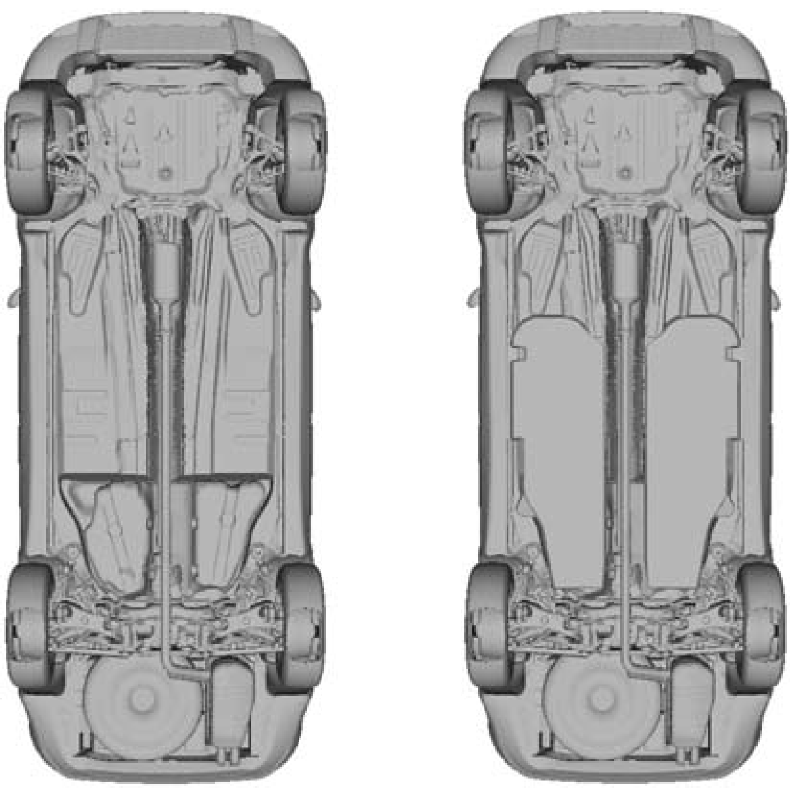

Figure 2: Bottom of the Volvo XC 90 without (left) and with (right) under body panel

Author

MSc
Zenitha Chronéer
is employee in the business area Environment and TASE of Volvo Car Corporation in Goteborg (Sweden).

Table 1: Editing methods of each module

Module	Editing method
Body/Exterior	Manual editing
Under body	Manual editing or reconstruction in Catia V5
Chassis	Surface Wrapping
Powertrain	Surface Wrapping
Wheels	Surface Wrapping
Brake system	Surface Wrapping
Under hood	Surface Wrapping
Fuel tank	Surface Wrapping
Cooling system	Manual editing

and tessellated surfaces. In the CFD group at Volvo, the vehicle is divided into modules. Before the vehicle is assembled, these modules are either cleaned, surface wrapped, or simplified.

2.2 CAD Data Clean-up

For the clean-up of the CAD data – simplifying, removal of gaps and intersections – different methods are used, dependent of the content of the module. The biggest time savings can be achieved by using surface wrapping. With this method the base geometry is shrinked with a surface mesh. Holes will be closed, intersections removed and, if needed, geometrical details can be removed. Theoretically, the whole model could be generated with surface wrapping. Practically, the meshes become too big and the numerous surfaces cannot be handled very well in Ansa. Usually, a mixture of surface wrapping und manual editing is used, **Table 1**:

- Body/Exterior: unnecessary details in the exterior surfaces are removed and gaps are closed manually with Ansa.
- Front and rear chassis, engine and wheels: Simplifing and correction with surface wrapping. The wrapper of CD-Adapco is executed in the batch modus via a fitting script
- Under body: generation of a continuous tessalated surfaces. Based on this construction of new surfaces in Catia V5. Hundreds of surfaces are reduced to ten.
- Cooling system: manual cleanup by the thermodynamic CFD team.

2.3 Model Assembly

The different modules are assembled using Ansa. Since octree-meshing is used, the structure doesn't need to be inspected for intersections and gaps. Only bigger holes with a size significantly bigger than the mesh size have to be considered.

2.4 Meshing

Harpoon was used fort the meshing. Harpoon is an octree mesher that produces a hex-dominant mesh. At the boundaries, the cut cells are converted into pyramids, tetrahedrons, or prisms in order to get the best quality. The computational domain is 50 m x 9.5 m x 9.6 m, **Figure 3**. For the vehicle structure the edge length is set to 5 mm. Only for critical domains, where separations were expected to occur, for example in the rear and on very fine details such as the bars of the grille, the size is set to 2.5 mm.

The maximum element size for the surrounding volume mesh was set to 320 mm. Around the entire vehicle, a refinement zone with size 40 mm was placed to assure that the cells do not grow too fast away from the surface. Mesh refinements were placed in the rear (20 mm), under the car (10 mm) and around the cooling package, to capture the wake, to resolve the narrow area to the ground, and to assure a constant mesh size within the cooling components (radiator, condenser, and charge air cooler).

The transition between differently dimensioned hex-zones was maintained by pyramids.

It is a conformal mesh, which means that transitions between two sizes of hexahedra are maintained by pyramids (conformal mesh). The final mesh consists of approximately 30 million cells and takes less than two hours to generate on a computer HP C8000.

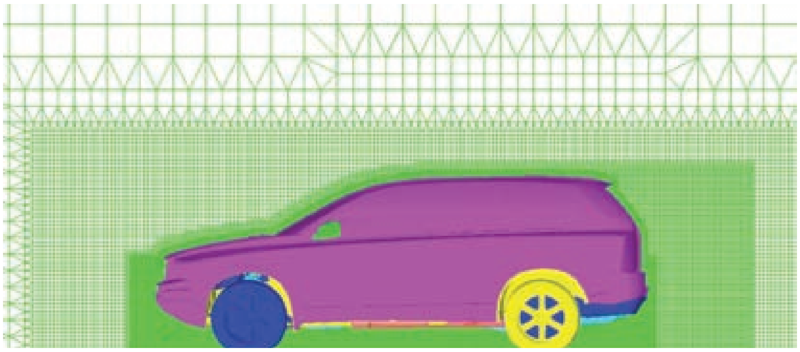

Figure 3: computational domain – cut plane through the mesh generated with Harpoon

3 Numerical Method

A pressure based coupled algorithm was chosen, which gives a robust and efficient solution for steady-state flows. For the convection terms in the momentum equations, a second-order upwind scheme was used while a first order upwind scheme was used for the turbulent properties. For turbulence modelling, the „Realizable kappa-epsilon Model" is used with standard wall functions.

The simulations are carried out on a Linux cluster of 32 CPUs (AMD Opteron 2.2 GHz). The typical run time is 36 hours for 2500 iterations. Convergence of the simulation is assumed when the residuals have decreased by at least three orders of magnitude and C_d shows a stable value or small oscillation (±0.001).

3.1 Boundary Conditions

At the entry into the computational domain a constant velocity of 140 km/h is set with a turbulence intensity of 0.1 % and a viscosity ratio of 200. At the outlet, zero gradients in the flow direction were specified for all variables. The ground is moving with the same speed as the free-stream flow, and the rotation of the wheels is consistent with the free-stream velocity. Fluids between the spokes of the wheels are rotating with the same velocity as the wheels. The cooling components are modelled as porous regions and are included in the solution via source terms. For the fan a specified pressure rise was set as a function of the velocity. The sides and the top of the domain are treated with symmetry boundary conditions.

4 Results

The simulations, done with the different meshes (ICEM CFD/TGrid and Harpoon) show in general a good correlation, but present clear differences in certain areas. Evaluating the quality of the Harpoon mesh, the results were compared with calculations on the base of a conventionally generated mesh: an Octree-mesh (ICEM-CFD) for the upper body, and a Delaunay mesh (TGrid) for the floor.

The ICEM-CFD/TGrid-Netz has prismatic layers on the upper body surface and on the wheels. The first prismatic layer has a height of 1.5 mm and an edge length of approximately 10 mm, which gives a y+ value of less than 100 for a velocity of 38.88 m/s.

Even Harpoon has the capabilities to make prismatic layers. But due to the relatively poor quality of the cells this option was not choosen. To get an acceptable low y+ value, a finer mesh was generated (cell size at 5 mm, in critical zones at 2.5 mm).

4.1 Effects of Under Body Panels

For both calculated variants (without/with UBP) the C_d values of the Harpoon mesh are by 0.016 below that ones of the ICEM-CFD/TGrid mesh. The difference of the C_d values between UBP and no UBP is 0.005 for both meshes, see **Table 2**.

Table 2: Value comparison for C_d without and with under body panels

C_d-value	ICEM/TGrid	Harpoon	Delta C_d-value ICEM/Harpoon
without under body panels	0.393	0.377	0.016
with under body panels	0.388	0.372	0.016
Delta C_d-value	0.005	0.005	

Bigger differences are visible at the c_l values, **Table 3**. In comparison to the testing results, both meshes predict the front lift too low and rear lift, **Figure 4**, too high. The c_l values for the Harpoon mesh are higher (0.109 without UBP, 0.086 with UBP). Even the difference with/without UBP is bigger for the Harpoon mesh. Responsible is the increase of rear lift.

Table 3: Value comparison for C_l without and with under body panels

C_l-value	ICEM/TGrid	Harpoon	Delta C_l-value ICEM/Harpoon
without under body panels	0.159	0.268	– 0.109
with under body panels	0.137	0.223	– 0.086
Delta C_l-value	0.022	0.045	

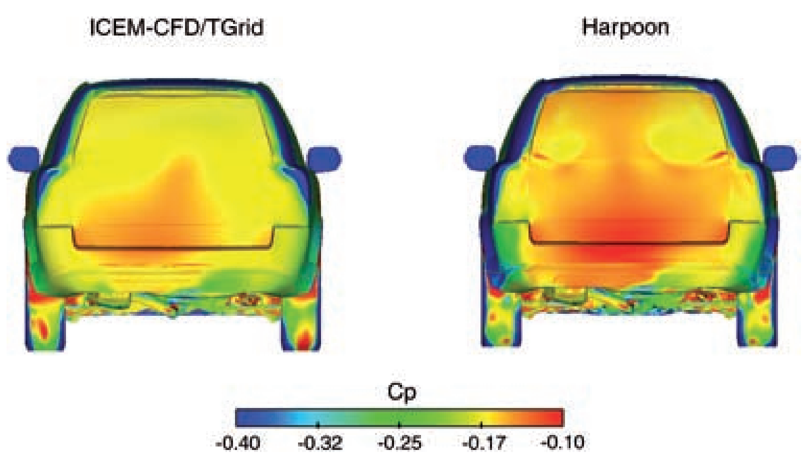

Figure 4: Pressure distribution on the vehicle rear (left: ICEM-CFD/TGrid mesh, right: Harpoon mesh)

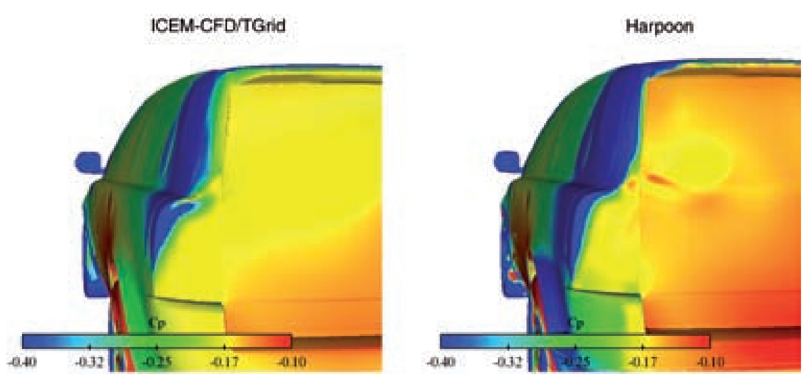

Figure 5: Pressure distribution on the rear lamp (left: ICEM-CFD/TGrid mesh, right: Harpoon mesh)

For the Harpoon mesh the pressure for the variant without UBP is slightly higher all over the base area. There is a larger region of low pressure on the rear lamp, **Figure 5**, for the ICEM-CFD/TGrid

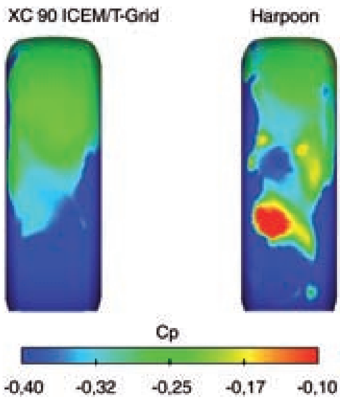

Figure 6: Pressure distribution on the rear side of the front wheel (left: ICEM-CFD/TGrid, right: Harpoon)

mesh compared to the Harpoon mesh. This indicates that separation occurs further downstream for the mesh made with ICEM-CFD/TGrid. Comparing to test results made in wind tunnel, the separation line for the Harpoon mesh looks more realistic.

Because of the difference in separation in the rear, the wake structure is different. The bottom half looks the same for the two meshes. But on the top, the Harpoon mesh looks more irregular. A possible explanation is a vortex that is formed over the cat walk, which is stronger for the Harpoon mesh.

Areas with a relative high pressure on the rear side of the front wheels, **Figure 6**, lower the C_d value in the Harpoon mesh. The part of the total C_d value that comes from the front wheels is 0.05 for the ICEMCFD/TGrid mesh and 0.03 for the Harpoon mesh. One reason for this difference could be the coarse mesh around the wheels in the Harpoon mesh. The y+-values (optimum 30–100) have an average of 200 significantly higher than fo the ICEM-CFD/TGrid mesh.

5 Post Processing

EnSight from CEI is used for the post processing. This process step was automated by using a script, which starts EnSight in batch mode and generates a set of standard plots. The plots show pressure distributions on the vehicle seen from different angles, velocities in different cuts through the volume, **Figure 7** and **Figure 8**, and iso-contours of a specified constant velocity. However, close-up images of details in the flow are made manually.

6 Conclusions

A CFD simulation for a completely new vehicle can now be carried out in four weeks. Updates on an existing model can be made within a few days.

The main ingredients for this process acceleration are a faster and error free geometry generation. In this context the collection of CAD data with TeamCenter and the usage of surface wrapping should be mentioned explicitly. But even the faster mesh generation with Harpoon was one of the main reasons for this speed up. Making a volume mesh of 30 million cells using Harpoon takes less than two hours on the computer HP C8000.

The carried-out calculations show that the results for the Harpoon mesh are comparable to the previous method, where the mesh was made by using ICEM-CFD (Octree) and TGrid (Delaunay). Since no prismatic layers were used with the Harpoon mesh, yplus values were fairly high. Therefore, the mesh size should be decreased on the wheels.

For the development process it is important, that both meshes show the same tendencies. This is essential, since often the comparison between different variants are much more important than absolute values. Before this background the obtained results and realizations can be rated only positively. ■

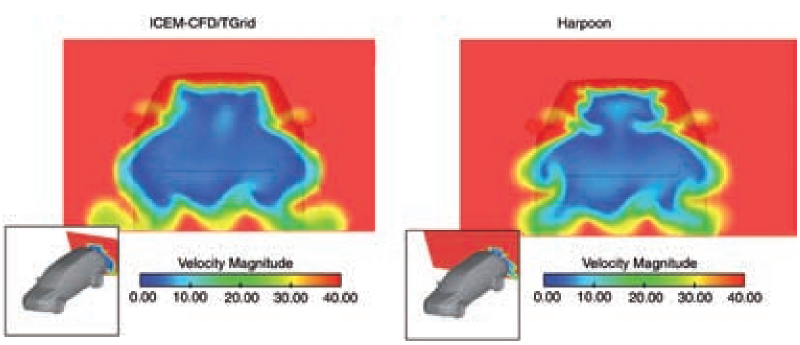

Figure 7: Velocity magnitude distribution in a cut plane 100 mm behind the vehicle (left: ICEM-CFD/TGrid, right: Harpoon)

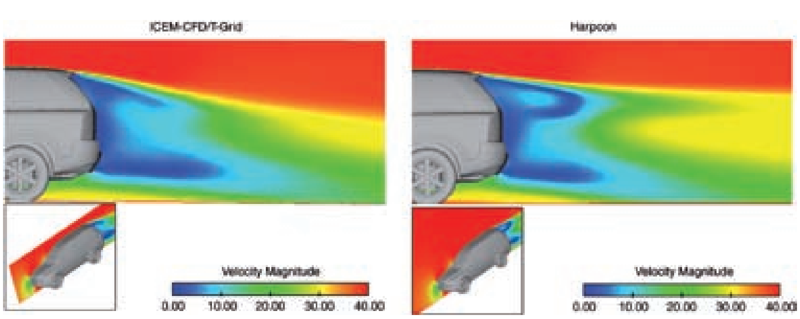

Figure 8: Velocity magnitude distribution in a cut plane through the volume at the centre of the vehicle (left: ICEM-CFD/TGrid, right: Harpoon)

Acknowledgements

Special thanks for their cooperation apply for Andreas Borg, Christer Bergström, Anders Jönson and Simone Sebben of the CFD group within the Volvo Car Corporation. Owing to also many thanks from the author at Mustafa Berispek for his cooperation, which led to the lowering of the turn-around times for aerodynamic simulations, as well as at Friedrich Bosch for his support concerning Catia V5.