AUTOMATED PRE-PROCESSING FOR HIGH QUALITY MULTIPLE VARIANT CFD MODELS OF A CITY-CLASS CAR

Evangelos Skaperdas*, Christos Kolovos
BETA CAE Systems S.A., Greece

KEYWORDS – CFD pre-processing, meshing, morphing, optimization, drag reduction

ABSTRACT

The power to efficiently run reliable CFD simulations is a great advantage for good aerodynamic design. This depends heavily on the ability of pre-processing tools to prepare and also be able to efficiently modify a high quality CFD model, overcoming all the difficulties that arise by the large model sizes and their complexity.

In this study the software ANSA v13 is used for the pre-processing of a medium size external aerodynamics CFD model. The focus here is on the application of the steps and techniques to prepare and modify such a model. The results verify the automation and quality achieved by the latest developments of the software. The model geometry is surface and volume meshed using the batch mesh tool, with minimum user effort, following all best practices for high quality models (curvature refinement, boundary layers, volume refinement zones, quality criteria checks etc), as applied in current industrial CFD applications. Surface wrapping approach is also followed and compared with the standard surface meshing of the batch mesh tool.

In addition, the integrated CAD tools and the Morphing functionality allow the CFD engineers to make their own design modifications when necessary, without any constraints or delays, in order to optimize a design. Here in particular, the effect of different inclinations and forms of the rear roof is examined with respect to the main aerodynamic characteristics of the vehicle, mainly drag and lift. The addition of vortex generators is also investigated to conclude if they can have an effect on such square back vehicle shapes. The resulting morphed design is output back to CAD systems in surfaces description.

1. INTRODUCTION

Computational Fluid Dynamics is an established tool in the aerodynamic development of passenger cars. Its main benefits, compared to real experiments, are lower cost and the fact that it provides a full picture of a complex flow, thus offering greater insight. CFD solver technology improves, and hardware performance increases continuously so more weight and investment is transferred to CFD, as nowadays very complicated turbulent engineering flows can be simulated with a high level of accuracy. The accuracy of the CFD analysis however depends also on the quality of the model that is built and sent to the solver. In addition, the whole potential of CAE simulation in the development process depends on the efficiency by which it can be completed. These are the challenges that a CFD pre-processor software has to confront.

The preparation of a mesh model for CFD analysis of an automotive application is a complex and demanding task, as both quality and efficiency must be combined [1]. The focus of the current work is to point out that, given a powerful pre-processing tool, the preparation of a high quality CFD model can be a straight forward and consistent process, with high user control but minimal actual effort. This is demonstrated through the use of the ANSA pre-processor for a test case of external aerodynamics of a mini city car model [2]. Although not the typical case for aerodynamic optimization study, the present work describes the full process to prepare the base model and later on to modify it in various aspects and examine the effect on its aerodynamics. This process is applicable to all current automotive needs. The modifications performed take place either by changing or adding geometrical constructions on the model, or by morphing of the final base meshed model. These modifications lead to seven variants of the base model. These include the addition of Vortex Generators at the rear end of the vehicle, subtle curvature modification of a front fender as well as more pronounced rear roof shape variations, all performed though the morphing capabilities of the software.

1.1. VORTEX GENERATORS

Vortex Generators (VGs) are devices that have been effectively used for quite some time in the aeronautical industry to improve lift and reduce drag of an aerodynamic surface, by delaying flow separation. The principle behind this is that these devices produce strong vortices that transfer momentum from the outer most regions to the lower regions of the boundary layer, thereby maintaining attached flow at positive pressure gradients.

More recently these devices are tested and placed on commercial cars and trucks. The work of Koike et. al [3] is a very good and validated example of the application of vortex generators on a sedan type vehicle aiming at reducing its drag coefficient, even by a small fraction. In such an application the vortex generators work as expected, that is they keep the flow attached along the downward slope of the rear windscreen and along the tail of the vehicle, leading to a narrower wake downstream and hence less pressure drag.

Some aftermarket VG devices also appeared, claiming to be designed for use on passenger cars and trucks in order to improve the aerodynamics of the vehicle with respect to drag, stability and rear visibility.
Such VGs have also been tested on trucks, installed between the tractor and the trailer and have demonstrated positive characteristics [4]. Another very interesting study [5] also shows some positive gain of VG installed on sedan passenger cars.
VGs installation is also suggested at the rear of a truck or a square back shape car. As this claim seems to deviate from the main principle of application of the VGs, the current study will try to evaluate this possibility.

2. GEOMETRY PREPARATION

Apart from the input of geometries through neutral CAD file formats like IGES, STEP and VDA-FS that can be read into ANSA, the mainstream approach of CFD pre-processing, as it is currently applied in the industry, is the direct translation of CAD data created in major CAD systems like Catia, Unigraphics and ProEngineer. This direct translation ensures optimum geometry quality input. In addition, all information, like part and property names and numbers, as well as assembly hierarchy, are also transferred, exactly as built in the CAD system. This allows easier management and manipulation of the model, as well as the ability to trace back a Part in the PDM system, if so required for some modification.

In this study however an alternative approach is followed. A tessellated representation of a small city car model is obtained from a commercial source [6]. This STL model consists of around 36 thousand triangular elements all belonging to a single Property. The model is not sealed as a watertight volume and the parts of the car are intersecting. It should be also noted that the STL file is a rough representation of a model and does not contain all the details or the accuracy of a real automotive application, like for example suspension, engine underhood details and has a flat underbody.
Using the available powerful CAD functionality, 3D Curves are extracted from the main feature lines of the model, as shown in Figure 1. Then using these feature lines as a wireframe, all surfaces are constructed. These created surfaces form also a fully sealed model without any intersections.

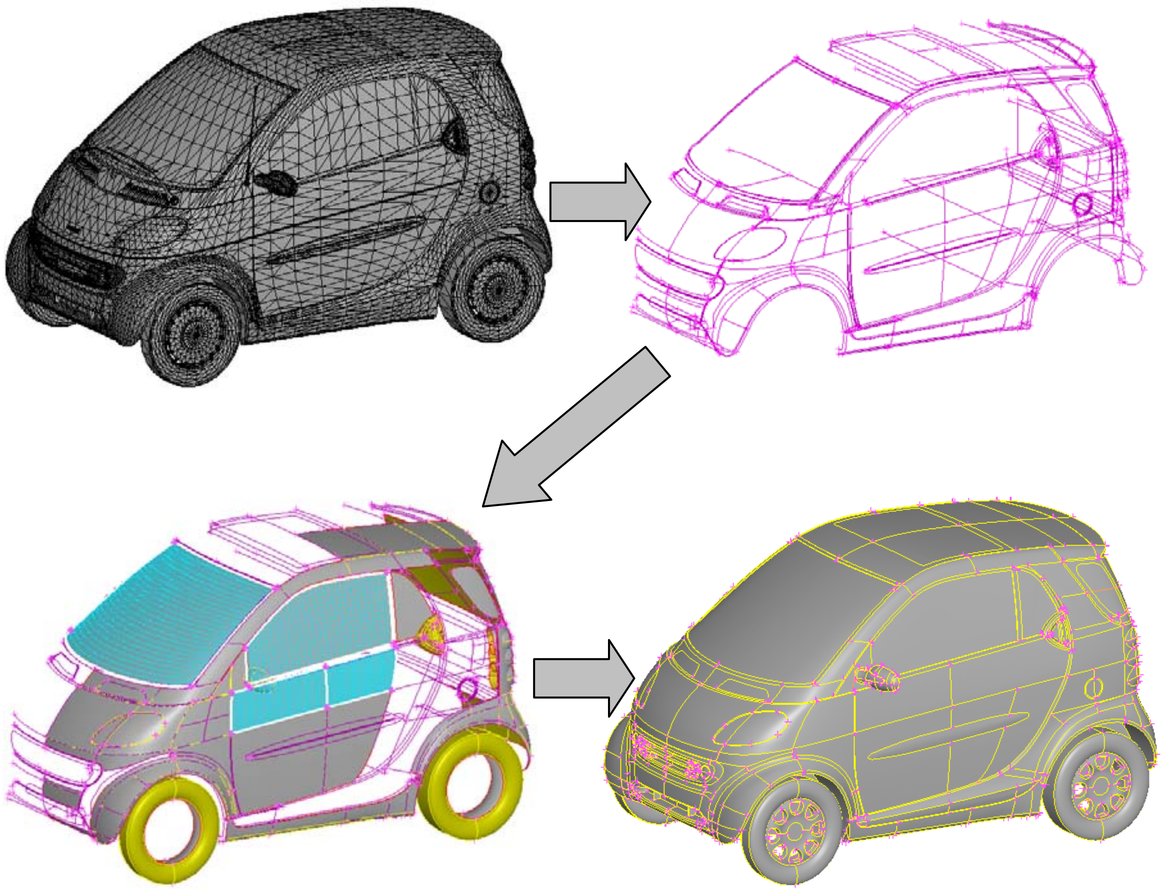


Figure 1: Geometry creation based on the STL input

The final model, along with the separated different parts is displayed in Property colours in Figure 2.

Figure 2: Final constructed model geometry

As the model does not have any non symmetrical features and a simplified flat underbody, to reduce the computational effort only a symmetric half computational model is created. A computational domain is constructed spanning 50m in length and 10 m in width and depth, as shown in Figure 3, resulting in a blockage ratio of less than 1%.

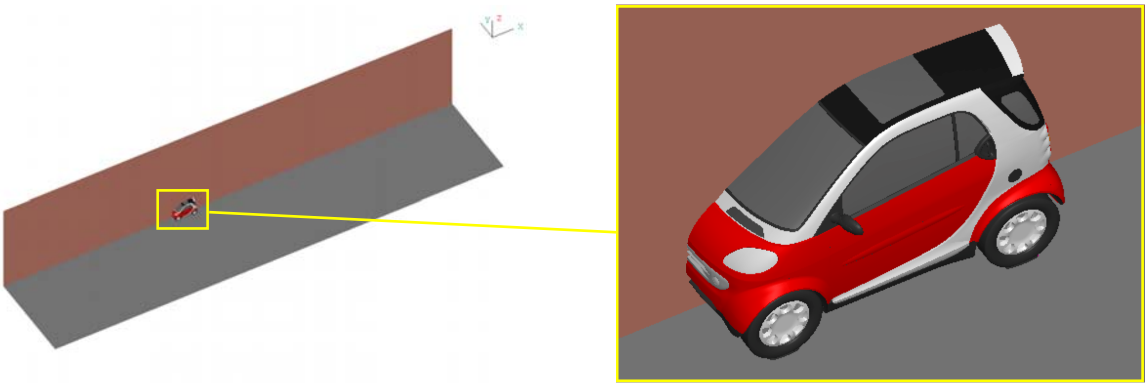


Figure 3: Computational domain and close up of car model

2.1. Addition of vortex generators

There are several Vortex Generator designs that have been tested in the literature and also available in the market, from simple thin triangular shaped surfaces to more elaborate designs. In this study the VGs are designed in the typical form of NACA type inlet profiles and are added on the base model. The dimensions of the vortex generators are shown in Figure 4, and have a height comparable to the boundary layer thickness in that area, as suggested in [3], for optimum effect.

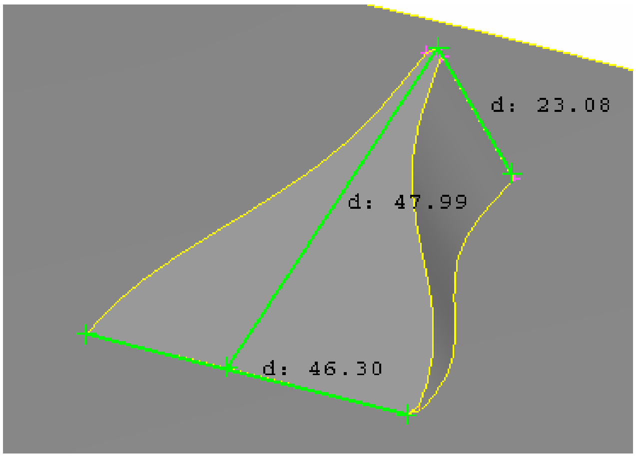
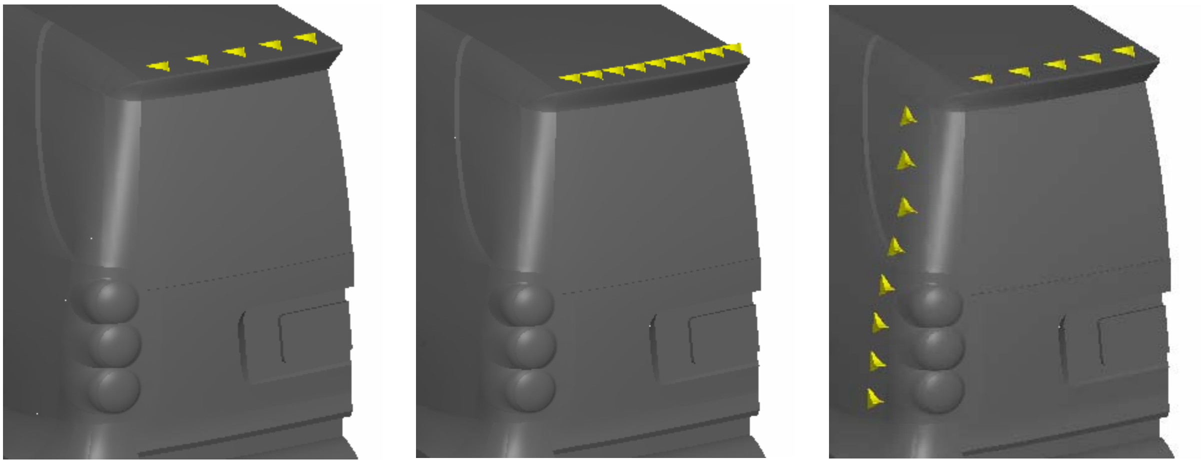


Figure 4: Close up and dimensions (in mm) of a VG

Three variants are built with respect to the positioning of the VGs on the model, as shown in Figure 5. The variations are on the interval spacing of the VGs as well as the location (top and side). Note that the VGs are not aligned to the local flow direction, but simply aligned to the longitudinal axis of the model.



VG1 VG2 VG3

Figure 5: Three model variants with respect to VG positioning (starting from left, VGs placed on top, on top dense, and on top and sides)

3. MESHING

The developments in the latest ANSA v13 have led to great enhancements in the automation of high quality CFD meshing. More specifically, starting from a clean and watertight geometry, the batch mesh tool is used to setup and run all the steps necessary for surface and volume meshing. There are three distinct steps; surface meshing, boundary layer generation and volume meshing. Each one is referred to as a scenario in the batch mesh tool. In each scenario, different areas of the model (vehicle and wind tunnel) are placed in different batch mesh sessions, each one with its own meshing specifications (element length, quality criteria etc.). These sessions also take in to account the refinement boxes that are placed inside the domain, in order to refine certain areas of importance even further.
Apart from the automation that the batch mesh tool offers for the generation of a CFD mesh for a specific model, another advantage is that each scenario is setup using property name filters which automatically distribute each part in the proper session. This allows the user to apply a consistent meshing strategy to any subsequent model, as long as some naming conventions are followed, by using the same batch mesh scenarios for every model.
All pre-processing steps took place on a Linux PC with two Quad Core Xeon CPUs at 2.6MHz and 32 GB of RAM.

3.1. Surface Meshing

For the surface meshing the automatic curvature-dependant meshing algorithm is used, where a maximum allowed mesh feature angle, and a growth rate, as well as a minimum and a maximum element length are specified. Sharp features of the model, which may also generate high gradients in the flow, are also automatically detected and refined down to a user specified length. In combination with the above local refinement boxes are created in order to further control the mesh in certain regions, like wake recirculation areas, separation lines and so forth. Specifically in this case, a base CFD run showed all these features and so size boxes are created to enclose them, as shown in Figure 6. Note that refinement boxes are also placed around the locations of the VGs to increase the mesh resolution. The same boxes are also applied to all the models under investigation, for mesh consistency.

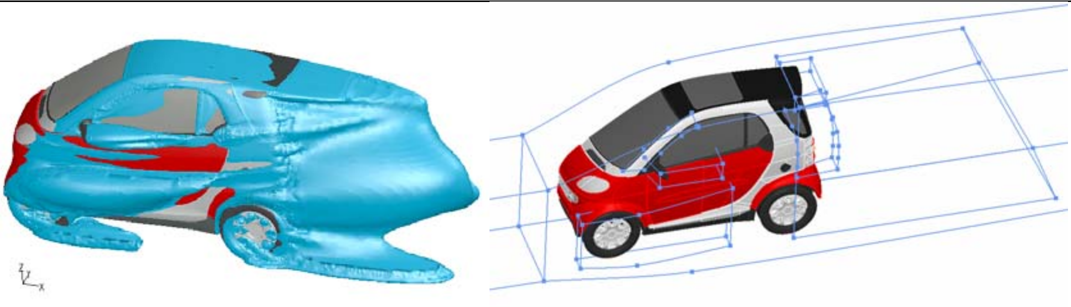


Figure 6: Iso-surface of total pressure equal to zero, as calculated in base CFD run (left) and final constructed Size Boxes for local mesh size control

Note that the refinement boxes do not have to be orthogonal. They can have any arbitrary shape in space and each one has its own max size value. This economizes further on the final cell count. A total of nine distinct size boxes, each one with its own maximum length, are constructed as shown in Figure 7. A box upstream of the car is also added to ensure an adequate number of cells between the inlet and the car, as suggested in [7].

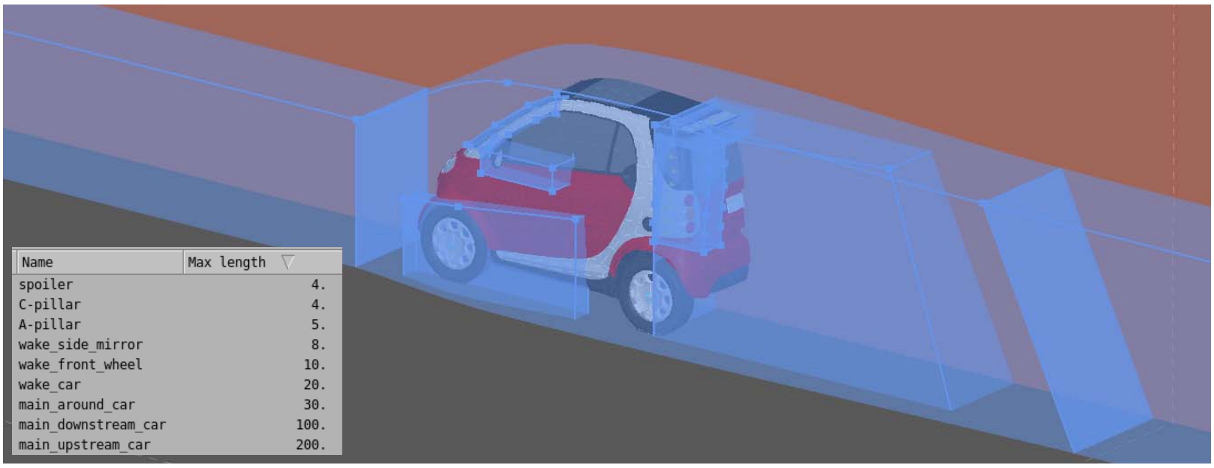


Figure 7: Size Boxes and corresponding maximum element length values

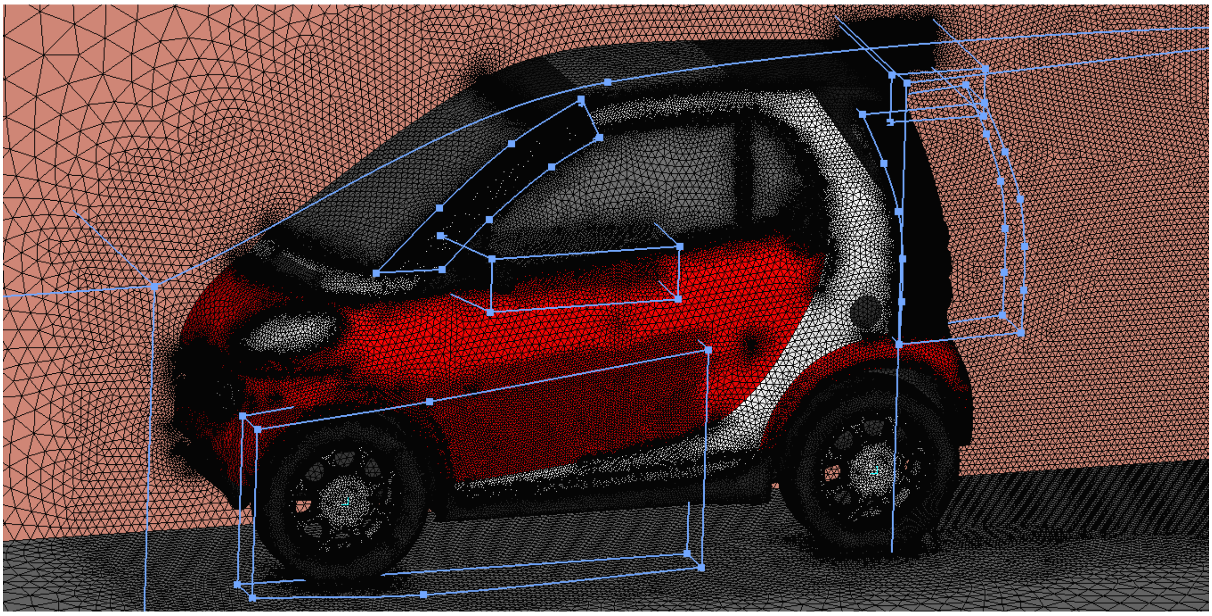


Figure 8: Surface mesh of approximately 452 thousand triangles, as created by batch mesh tool

The resulting surface mesh, consisting of approximately 450 thousand triangles (Figures 7 and 8) is a very high quality, resolving all important features of the model, while keeping the element count low in flat areas. The mesh size growth results in a very smooth transition with a growth rate factor of 1.2. With the use of the batch mesh tool, this scenario requires less than five minutes to complete.

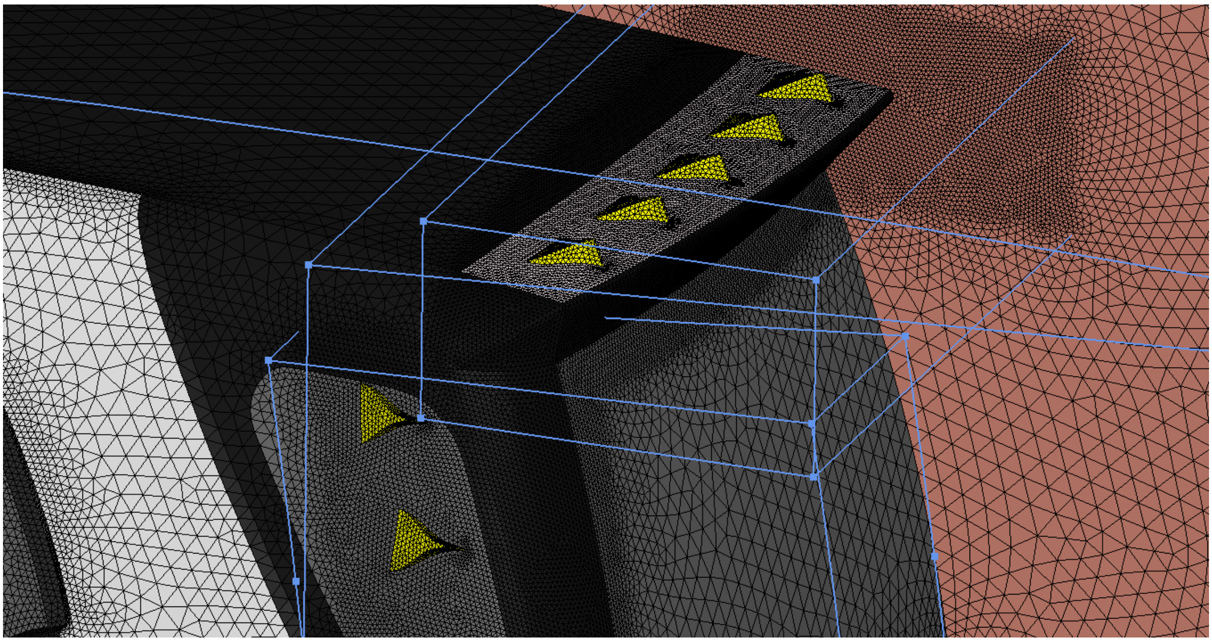


Figure 9: Close up of the surface mesh at the rear spoiler area showing the VGs

3.2. Surface Wrapping

The need for shorter time cycles of CFD model preparation has lately led to a more widespread approach of surface wrapping. The benefit of this approach is of course the fact that there is no need to prepare a fully watertight model, the most time consuming step of a CFD model build up. The same geometry that was surface batch meshed was also fed to the new Surface Wrapping tool of ANSA v13. The tool captures all the important geometrical features of the model, refining the curvatures and also the areas inside the same Size Boxes. The result is a watertight surface mesh, shown in Figure 10, prepared in under 5 minutes.

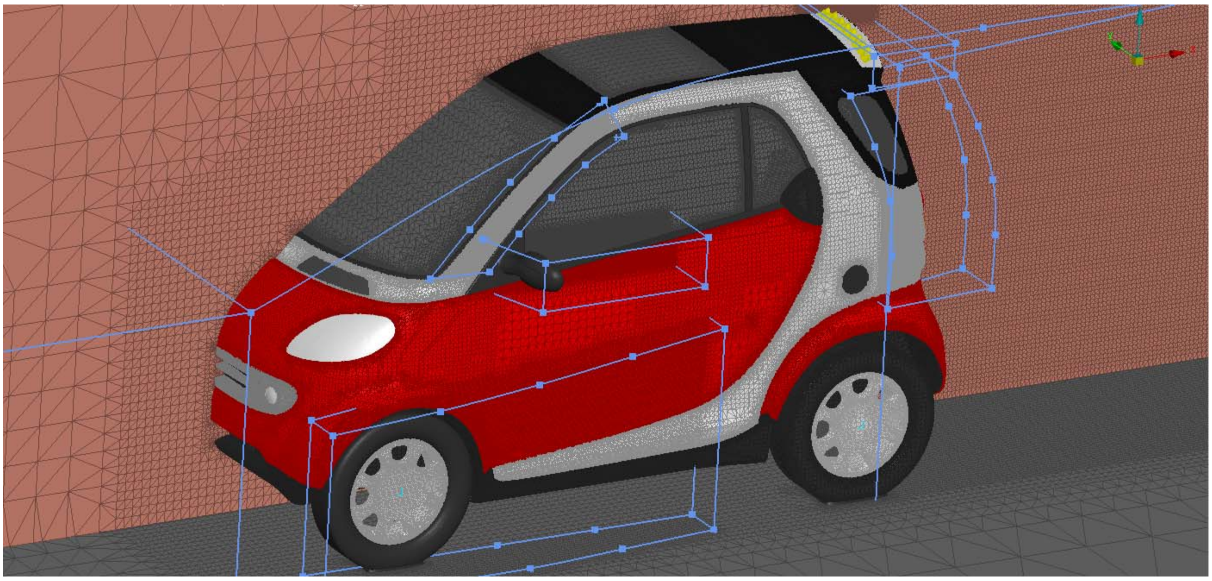


Figure 10: Surface Wrapping mesh

The result of the surface wrapping is then sent for quality improvement leading to the mesh shown in Figure 11. Comparing the standard surface mesh model (Figure 8) and the surface wrapping mesh (Figure 11) one can observe that both attain a high level of quality. This is not to say of course that the standard surface meshing approach offers the best quality and user control.

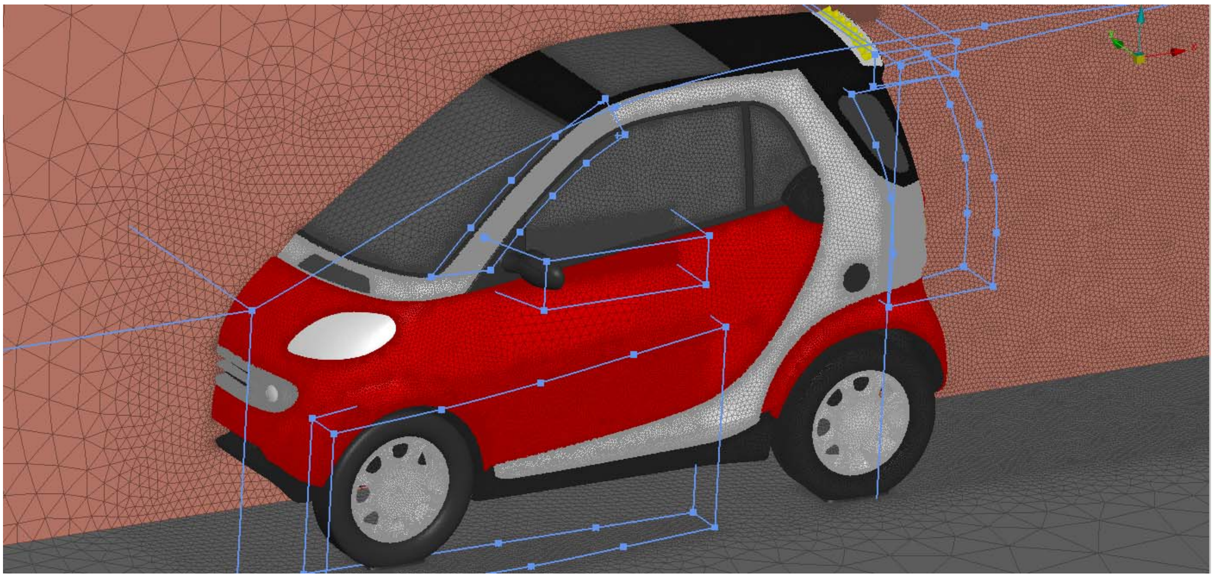


Figure 11: Surface wrapping mesh after quality improvement step

3.3. Layers generation

For such an analysis a volume mesh with boundary layer elements is of great importance. A batch mesh scenario for the layer generation is here setup.

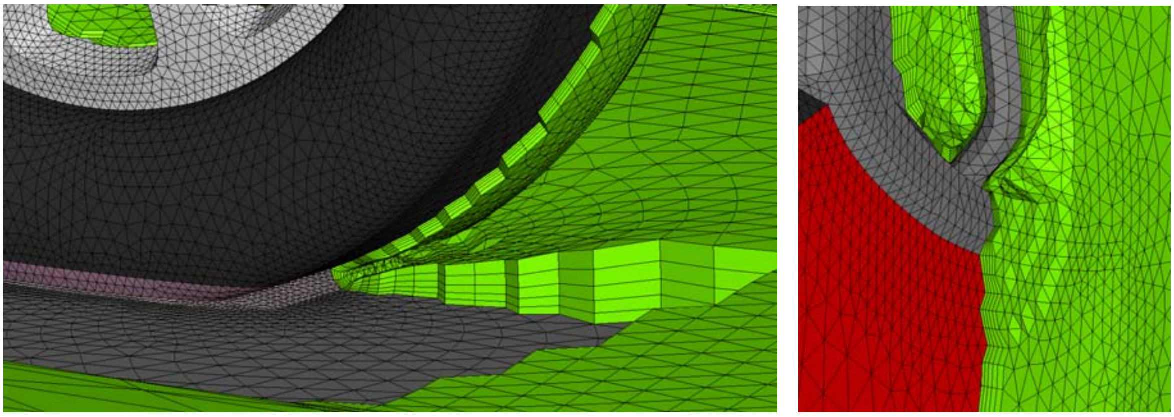


Figure 12: Close up of the layers near the tyre and road and around the door handle

Generation of boundary layers is a highly controllable and robust process. Layers are generated with different growth settings for different areas. Five layers are grown with a geometrical growth rate of 1.2 from the car and wheels with an absolute first height of 1mm, while from the road they grow in aspect mode with a first height of 20% of the local element length. This mode for layer creation is selected for the road only as it gives a much better aspect ratio of pentas and size variation with the remaining volume mesh away from the car where the element length increases to 500mm.

The option of layer squeezing to avoid intersection and proximity problems is also employed. Squeezing of layers is limited down to a minimum length of 0.3mm. If this condition cannot be respected then the generation of layers automatically excludes such problematic areas which are then filled with tetra mesh. The connectivity between the layer sides and the tetras can be achieved through pyramids or the creation of non-conformal interfaces. Figure 12 shows the layer squeezing effect and the different layer parameters between vehicle and road. In Figure 12 a cut section of the layers is shown around the VGs. The layers connect automatically to the remaining mesh of the wind tunnel boundaries and the symmetry plane. All the above parameters are setup in the batch mesh tool as a Layers scenario and executed automatically. The final layers consist of approximately 1.9 million penta elements meshed in under two minutes.

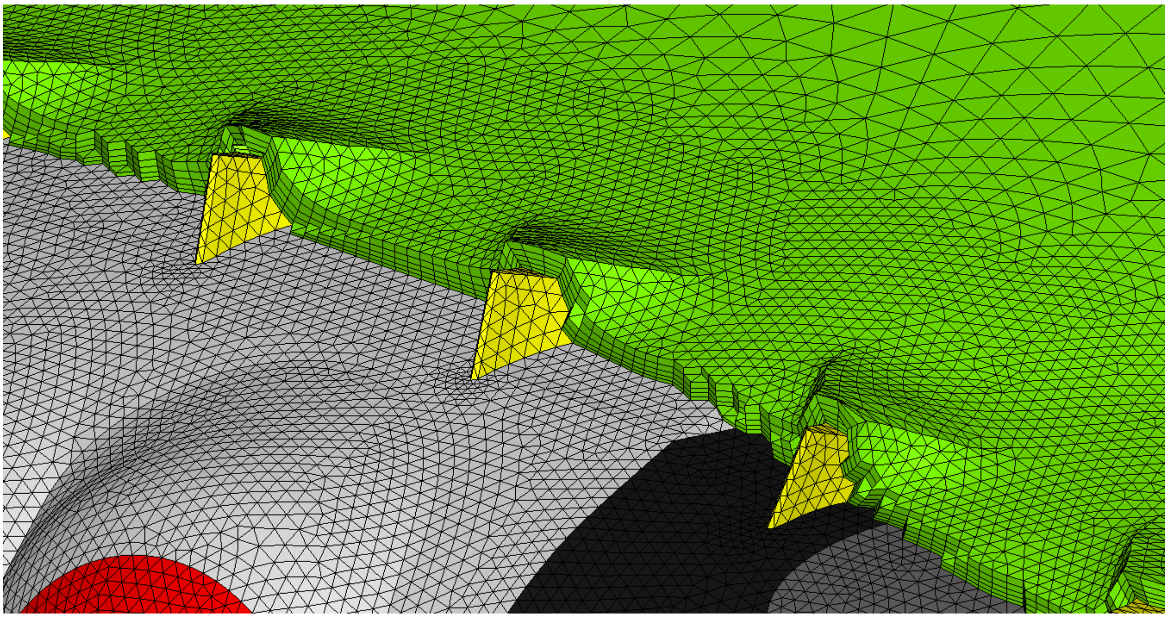


Figure 13: Close up of the layers around the side VGs

3.4. Volume mesh

The Hexa-Interior mesh approach is followed to reduce the cell count in the large bulk refinement areas. A pure tetra mesh leads to 22 million tetras (Figure 12), whereas the Hexa-Interior mesh creates around 9 million for approximately the same resolution.
The Hexa-Interior mesh created is of variable size, adjusting to the local surface mesh detail, and is a fully conformal mesh without any hanging nodes. The same refinement boxes that are used to refine the surface mesh, also apply to the volume mesh, thereby giving perfect match in size.
Volume meshing is the third scenario in the batch mesh tool sequence. It automatically detects the remaining volumes, after the generation of the layers, and meshes them with the user predefined parameters for maximum length and quality thresholds. The created volume mesh, consists of 9 million hexas, tetras, prisms and pyramids and is completed in approximately 15 minutes.

The required memory for the whole final model, shown in Figure 15, which including the layers consists of 11 million elements, requires around 5Gb of RAM during its generation. Regarding the quality, for the surface mesh a max Equiarea skewness of 0.5 is kept, while for the volume mesh an Equivolume max skewness of 0.95.

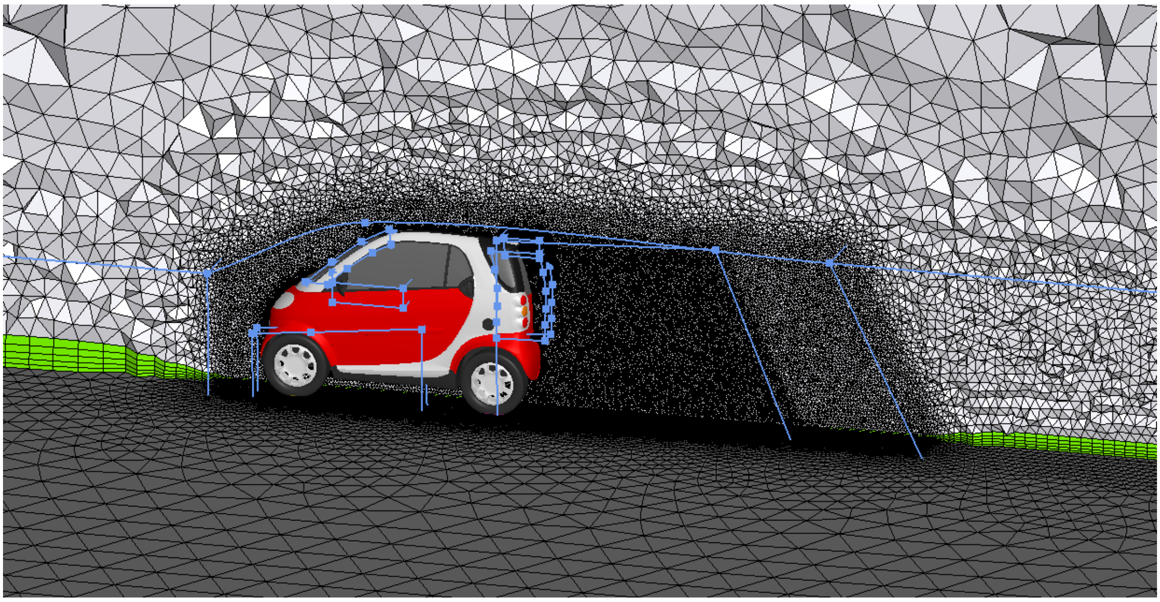


Figure 14: Section of the tetra mesh with the Size Boxes – 22 million elements

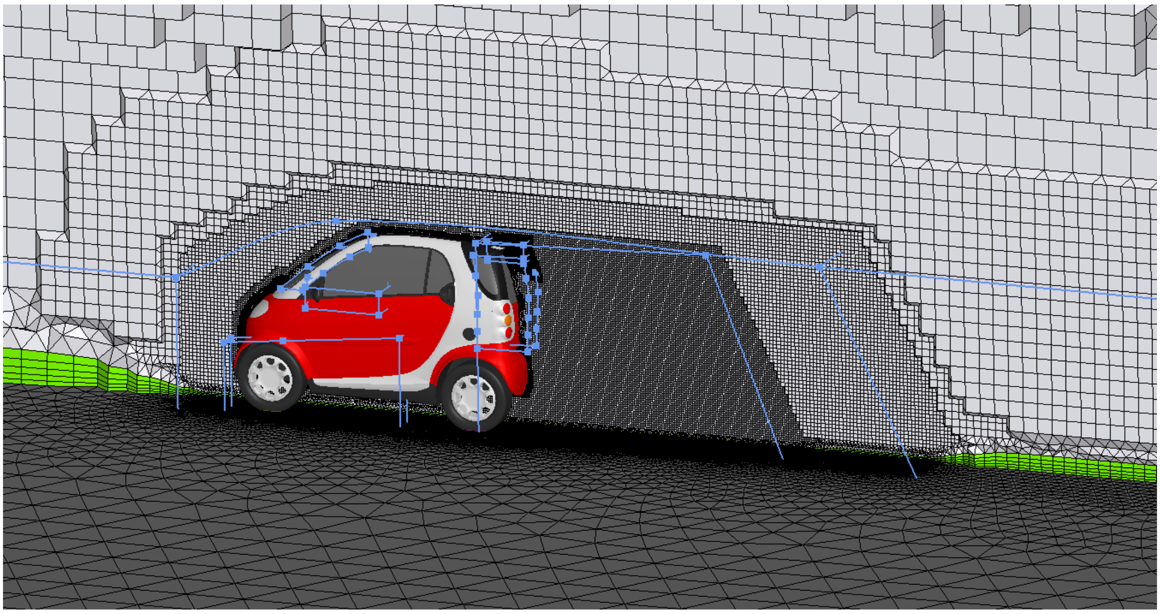


Figure 15: Section of the Hexa-Interior mesh with the Size Boxes – 9 million elements

4. MODEL MORPHING

Morphing can be performed very easily and with high control for a complete meshed model (surface and volume mesh), as well as for CAD surfaces. For optimization studies the morphing of the meshed model is preferred, for speed and simplicity of handling.
The morphing functionality is based on Morphing Boxes which contain the area of the model to be reshaped. Splitting of the boxes and local snapping of their edges on the model feature lines, as shown in Figure 16, allows very precise morphing of the model. Note that the domain that the boxes occupy extends quite far from the area that is actually morphed. This ensures that the mesh “strain” is minimal and the quality is not compromised. Starting with a high quality mesh also helps of course as it increases the extent by which the mesh can be morphed without causing quality problems. Note that the morphing boxes are like templates and once setup can also be used on other models.
In this study Morphing is used to change the curvature of the front fender and the rear roof inclination and spoiler setting. The morphing boxes are constructed on the surface mesh

model. This allows easy and quick preview of the applied deformations. The preparation time for the boxes is less than 15 minutes.

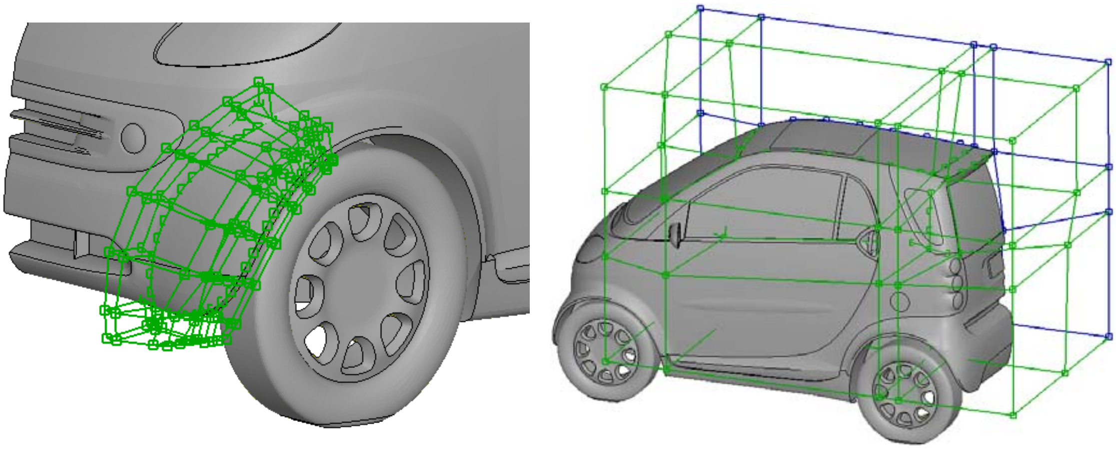
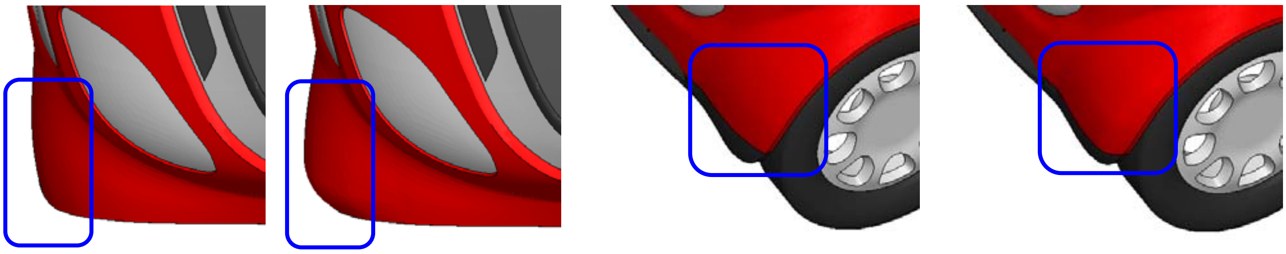


Figure 16: Morphing Box templates for the modification of the front fender (left) and the rear roof (right)



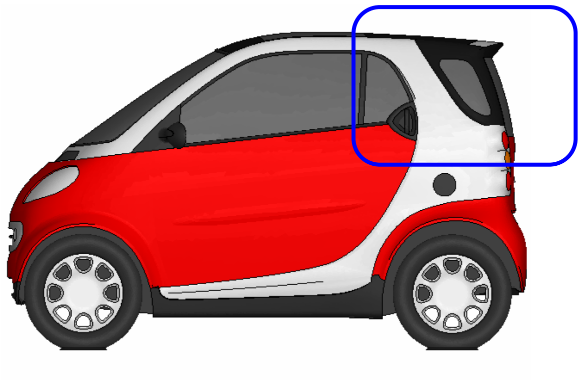
base M1 base M1

Figure 17: Morphing of the front fender to alter its curvature, top view (left) and isometric view (right)

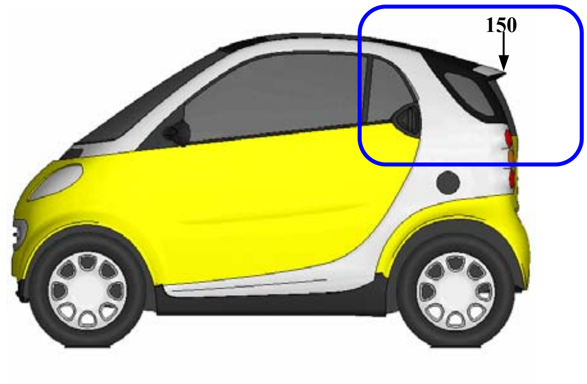
The morphing process is instantaneous and as many variants as required can be quickly output. Figure 17 shows the curvature variation of the front fender by approximately 25mm at the maximum deformation area.
Figure 18 shows the three variants for the rear roof end. The boxes snap the spoiler at its start and end, thus allowing its repositioning in height and also inclination. The final morphing state can be obtained after a simple movement of control points or a combination of control point movements. In any case, the original and final states are recorded and both, including any interpolated state in between, can be retrieved immediately. The morphing boxes, which contain also the information of the original and final states are exported separately and merged to the database with the full volume mesh. Then the same morphing is applied to the volume mesh in a single step as shown in Figure 19.

All morphing actions can be parameterized and hence the morphing tool can be coupled with an optimizer software for a fully automated optimization loop study as demonstrated in [8].

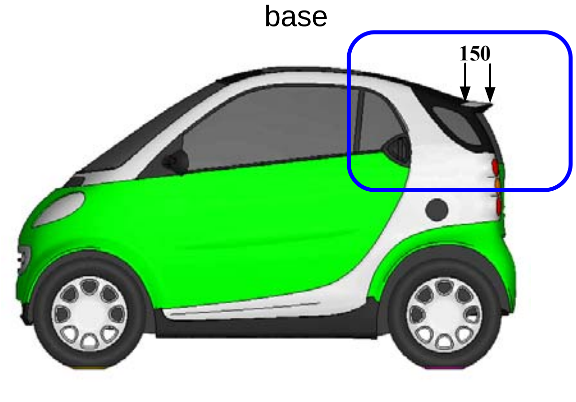
In the end, the final morphing is applied to the CAD surfaces, as shown in Figure 20, using the same Morphing Boxes and recorded history, so that the optimum geometrical description can be sent back to the CAD department.



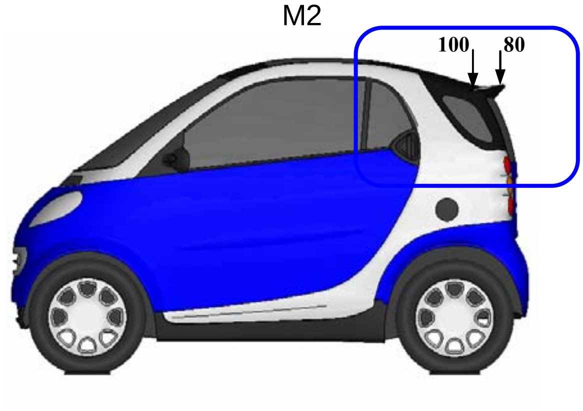
base



M2



M3



M4

Figure 18: Imposed deformations, in mm, on the rear roof end for the three variants (Red is base model)

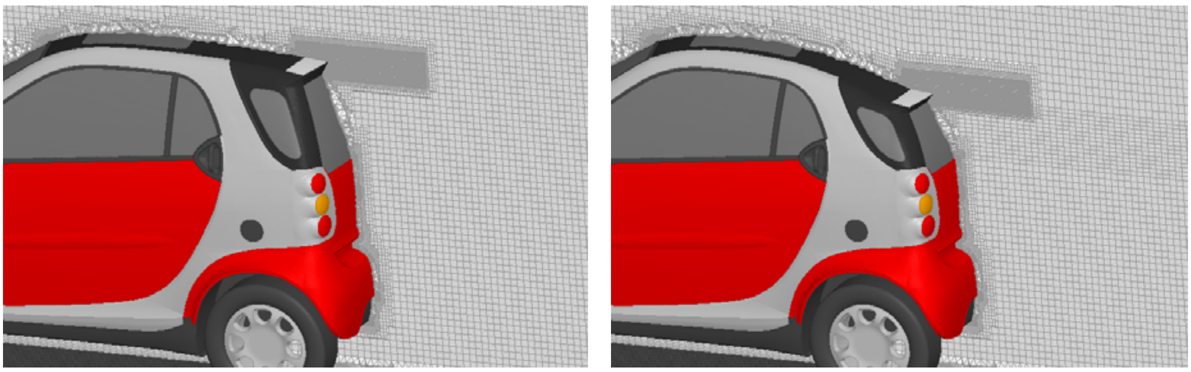


Figure 19: Morphing of the volume mesh model for M3 variant

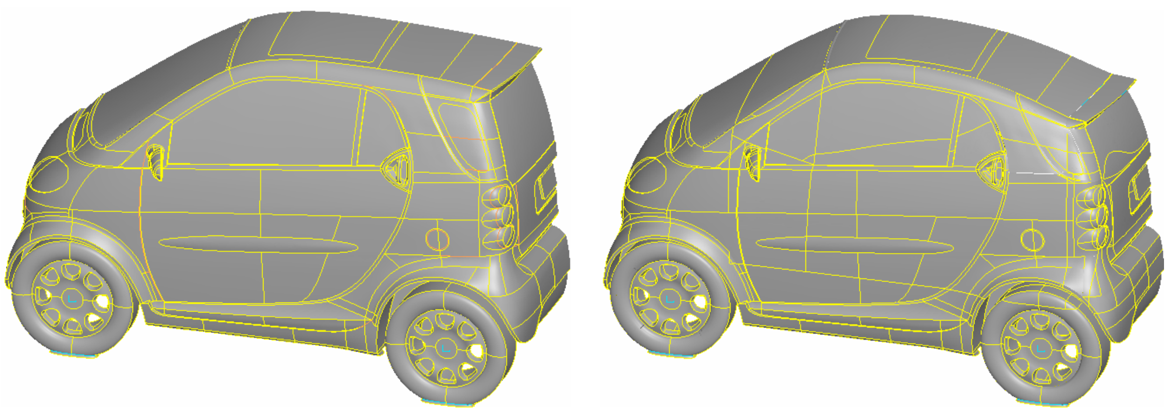


Figure 20: Original and morphed geometrical data for M3 variant

5. CFD SIMULATIONS

The simulations are performed using Fluent 6.3.26 [9] on the same Xeon PC configuration. Steady state flow is considered. The k-ε realizable turbulence model is employed. All equations are discretized with a second order numerical scheme.
A uniform velocity inlet of 38.89m/sec is specified, with a turbulence intensity of 0.1% and a viscosity ratio of 200. The road is moving with the same free stream velocity and the outer walls are set as symmetry type. Residuals, drag and lift coefficients are monitored until full convergence after around 3000 iterations.
The created layers ensure a suitable range of y^+ for most of the model, as shown in Figure 21.

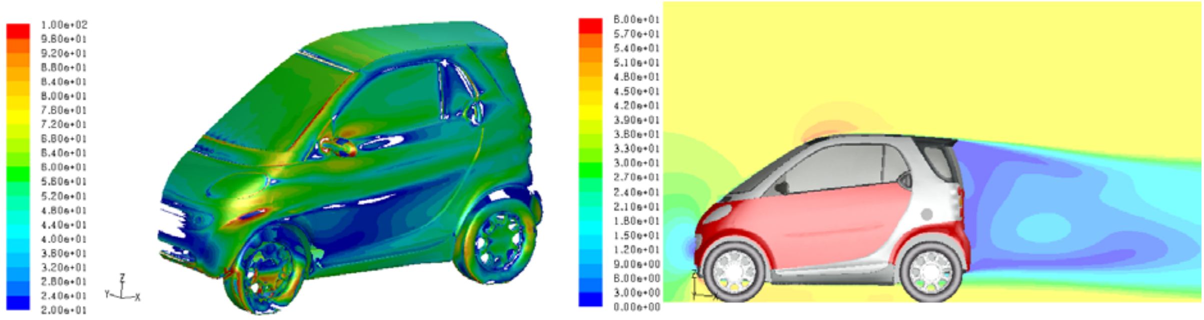


Figure 21: Contours of y^+ in the range of 20 to 100 (left) and velocity magnitude contours on the symmetry plane (right) for base model

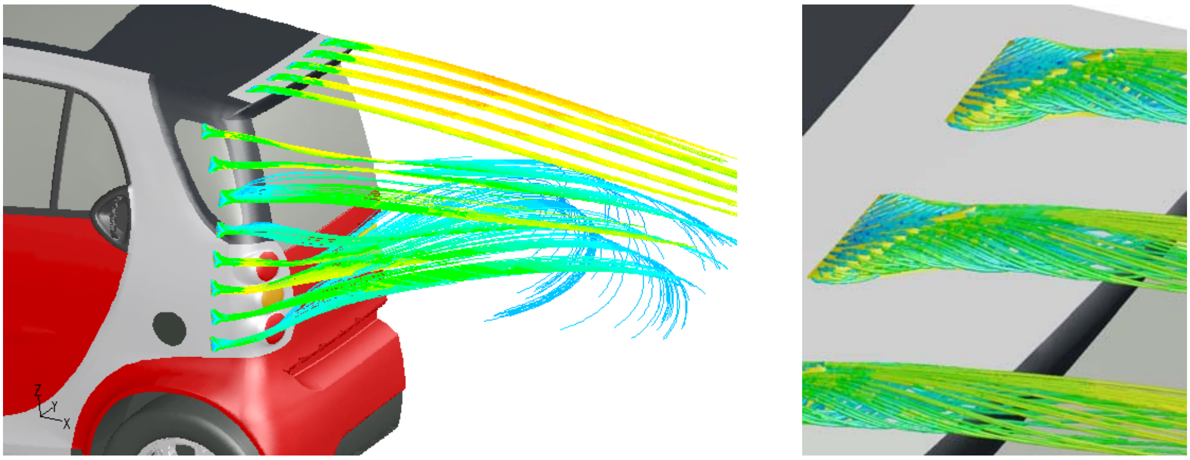


Figure 22: Streamlines coloured by velocity magnitude, released from the VGs for model VG3 and close up of the vortex formation around them

A typical flow field for a square back vehicle is observed in Figure 21, with a large wake and an absence of horse shoe counter rotating vortices that are generated by inclined C-pillars in other vehicles [10]. The traces of the VGs can be shown clearly in Figure 22. However these vortices do not seem to have a pronounced effect on the flow. In fact the drag coefficient is marginally increased as more VGs are added, as they consume more energy from the flow.
A summary of the results can be found in Figure 23, in the form of static pressure coefficient at the rear side of the vehicles, velocity magnitude and turbulence intensity contours 150mm downstream of the spoiler, as well as iso-surfaces of total pressure equals zero to indicate the areas of losses. It can be observed that the effect of VGs is minimal and much localized. The upper vortices are stronger as the VGs that create them are exposed to a cleaner flow. The sides are exposed to disturbances generated from the front wheel and the door handle.

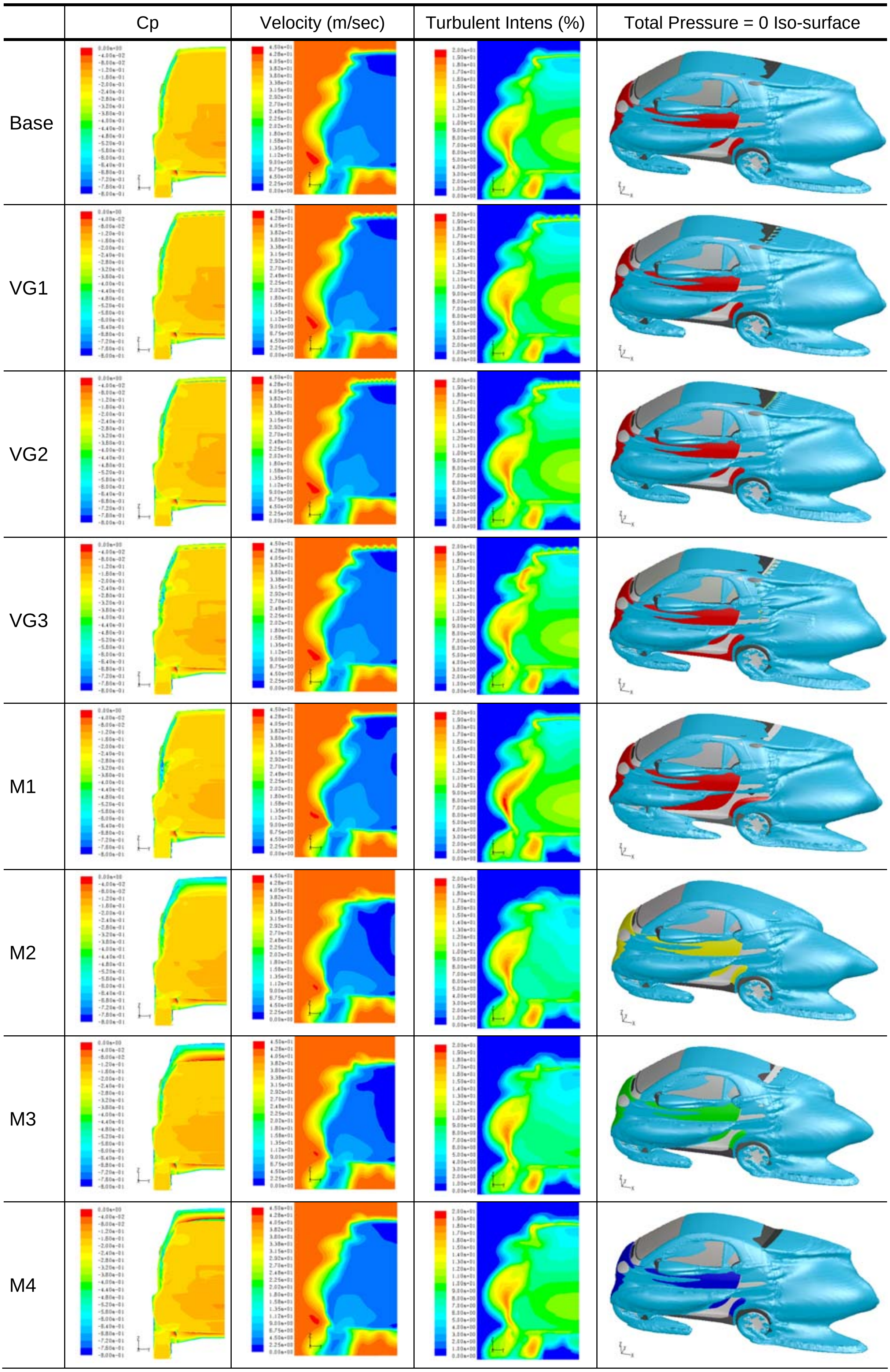


Figure 23: Summary of results for the eight model variants

The morphed models show more pronounced flow differences. The small change in curvature of the front fender for model M1 seems to alter significantly the flow field around and downstream of the front wheel
For models M2 to M4 there are more pronounced changes of the developed flow. Although the rear base pressure is not altered, the size reduction of the recirculation area results in a reduction of the total pressure drag for these models.

6. CONCLUSIONS

The complete pre-processing functionality and batch mesh automation available in the latest ANSA v13 allows the creation of a complete external aerodynamics CFD model of approximately 11 million volume elements in less than 25 minutes, without any user intervention, starting from a clean unmeshed geometry. The resulting mesh is of high detail and quality following all best practices for element quality, boundary layers and refinement zones. The whole process is automated and robust and can be re-applied on subsequent models ensuring consistency and time reduction of CFD model preparation. This takes the tedious part of the CFD simulation work away from the engineer and allows them to focus more on the actual flow investigations. The surface wrapping tool provided in ANSA v13 demonstrates a great potential for the creation of high quality CFD meshes with minimum effort.
The available morphing functionality offers efficient generation of several variants from a single base model. The initial high mesh quality allows the deformation of the mesh to significant extent without the need for model remeshing.
Morphing of the final model can be applied on the geometrical model with the same ease, so that the optimized surfaces can be sent back to the CAD department.

The study shows that, at least for the tested configurations, the addition of Vortex Generators at the rear of this square back vehicle had insignificant effect of the flow. In fact the drag coefficient is marginally increased as more VGs are added, as the additional flow energy that they consume is not counter balanced by any positive effect.

REFERENCES

[1] E. Skaperdas, C. Kolovos, “Combining Quality, Performance and Efficiency in CFD Pre-Processing”, Proceedings of the 2nd ANSA and META International Congress, Halkidiki, Greece, 2007
[2] E. Skaperdas, C. Kolovos, “Étude in CFD Pre-Processing”, Proceedings of the 7th MIRA International Vehicle Aerodynamics Conference, Coventry, UK, October 2008
[3] M. Koike, T. Nagayoshi, N. Hamamoto, “Research on Aerodynamic Drag Reduction by Vortex Generators”, Mitshubishi Motors Technical Review No. 16, 2004
[4] “AIRTABS™ Vortex Generators Technical Trials Articulated HGV”, IRTE/BTAC Technical Trials at MIRA Report, 1994
[5] “Blowing the Vortex”, www.autospeed.com/cms/A3058/article.html
[6] www.the3dstudio.com
[7] M. Lanfrit, “Best Practice Guidelines for Handling Automotive External Aerodynamics with FLUENT”, Version 1.2, http://www.fluentusers.com, 2005
[8] F. Campos, P. Geremia, E. Skaperdas “Automatic Optimization of Automotive Designs using Mesh Morphing and CFD”, Proceedings of the 3rd EACC, Frankfurt, Germany, July 2007
[9] Fluent v6.3.26 User’s Guide, Fluent Inc
[10] W.H. Hucho, “Aerodynamics of Road Vehicles”, 4th Edition, SAE International, 1998