ExaFLOW use case: Numerical simulation of the rear wake of a sporty vehicle

C. J. Falconi D.[1], D. Lautenschlager[2]

[1]*Automotive Simulation Center Stuttgart e. V.*
Nobelstraße 15, 70569 Stuttgart, Germany

[2]*Adam Opel AG*
Friedrich-Lutzmann-Ring, 65423 Rüsselsheim, Germany

1. Introduction

The test case under consideration is the airflow around the vehicle OPEL Astra GTC. This vehicle was designed by the automobile manufacturer Adam Opel AG. Additionaly to windtunnnel testing this vehicle was aerodynamically developed by using Computational Fluid Mechanics (CFD) applying the Reynolds-averaged Navier–Stokes equations (RANS) for the full detail vehicle model. For a more detailed understanding of the behavior of turbulent structures Detached Eddy Simulations (DES) were used for simplified submodel domains. The Reynols number is equal $Re=6.3\times10^6$ using the wheel base (L=2.695m) as the characteristic length and the INLET velocity (140km/h) as reference velocity.

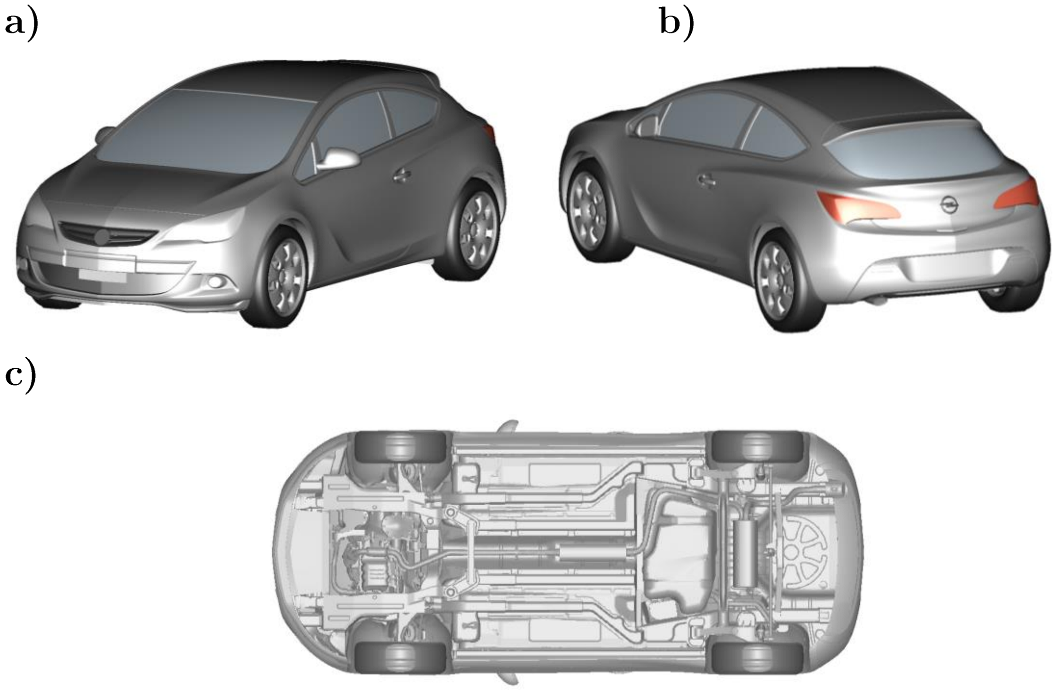

Figure 1: Geometry of the Opel Astra GTC. a) Front top view, b) Rear top view, c) Bottom view.

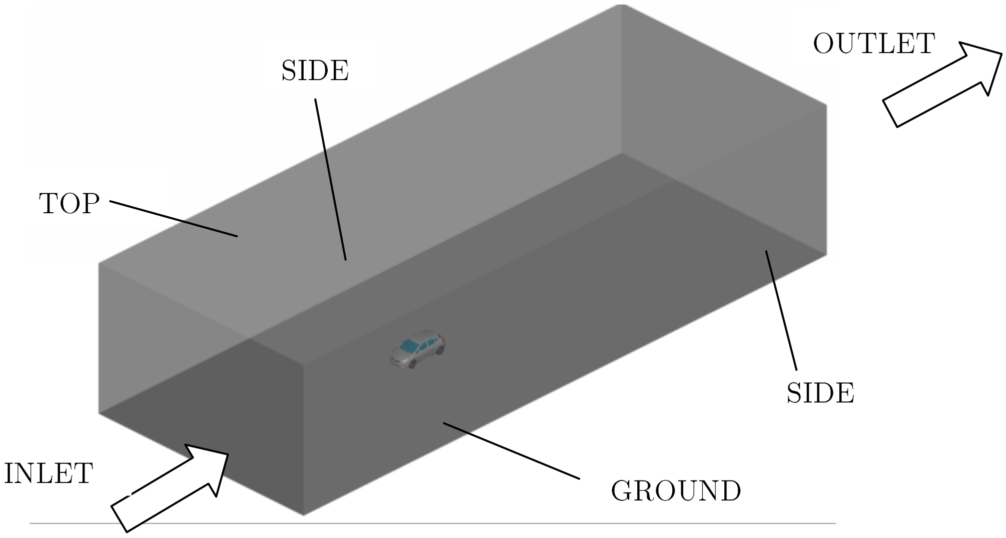

Figure 2: Computational domain of the completely vehicle model in the wind tunnel.

The respective simulations were initially performed with the commercial software ANSYS Fluent version 6.122 during the development process of the vehicle. In order to provide the first requirements and description of the automotive use case in the frame of the Exaflow project the simulation results were updated by the use of ANSYS Fluent 14.0.

After describing the geometry of the vehicle in section 2, the complete vehicle model will be described in section 3 followed by the description of the so called submodel in section 4. In the last section the need of exascale capabilities for the use case is specified.

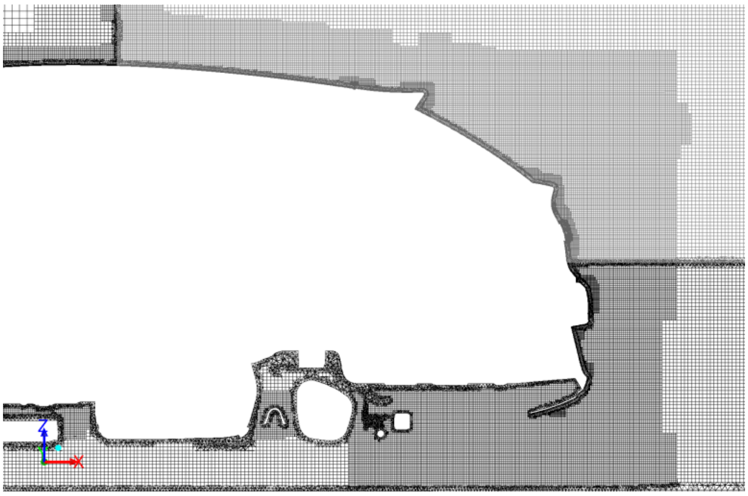

Figure 3: X-Z plane of the numerical mesh used for the completely vehicle model at the middle of the vehicle

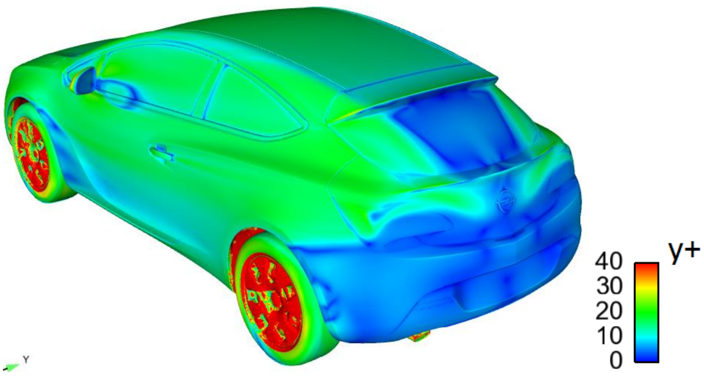

Figure 4: Distribution of the non-dimensional wall distance y^+.

2. Vehicle geometry

The Opel Astra GTC is a 3-door sporty looking vehicle; the vehicle is 4.468m long, 1.991m wide and 1.449 m height (see Fig 1a, 1b, 1c). In order to ensure a successful mesh generation process a high quality CAD model is required. The engine compartment and the underbody are modeled in full detail in order to capture all relevant aerodynamic effects.

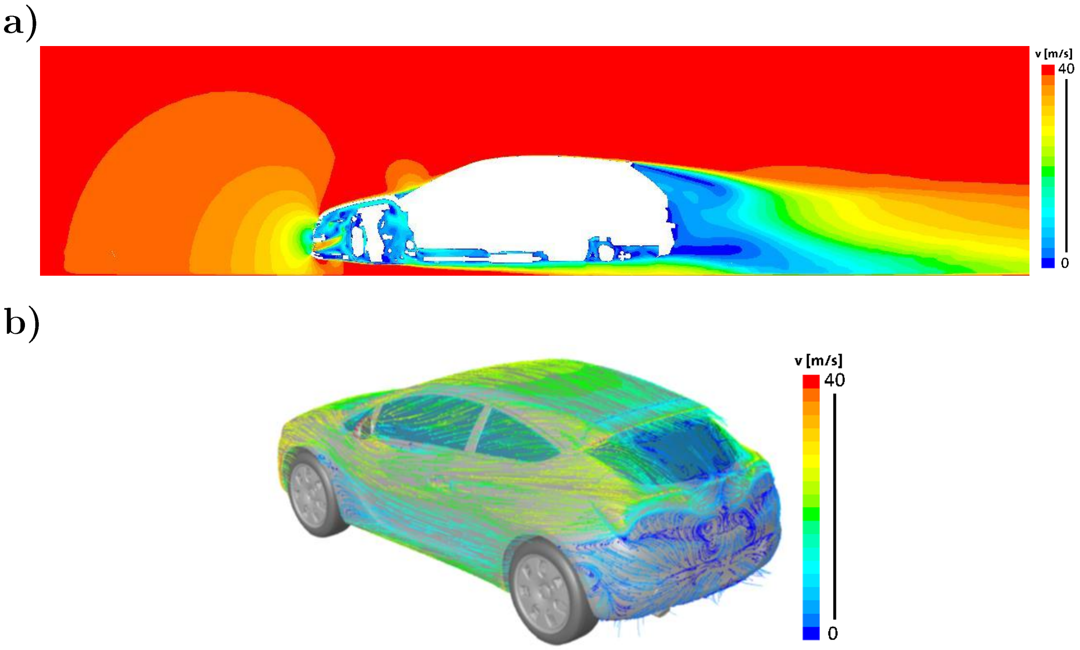

Figure 5: Simulation results *(RANS)* of the complete vehicle model. a) 2D contour of the velocity magnitude at the middle of the vehicle. b) Streamlines colored by the velocity magnitude.

3. Complete simulation model

The complete simulation model consists of the entire vehicle geometry described in section 2 and the virtual wind tunnel as depicted in Figure 2. The wind tunnel is 51m long, 20m wide and 12m high.

3.1 Numerical method

Simulations were performed using the Reynolds-averaged Navier–Stokes equations (RANS). The turbulence effects were modelled by the realizable k-ε model. The numerical mesh on the surface of the vehicle was generated with the commercial mesh generator ANSA. The typical length of the elements on the external surface is between 1-6mm, while surfaces near the underbody region were meshed with a length between 2-12mm. The total number of elements of the completely 2D surface mesh is 3.4 million.

Based on the surface mesh, a 3D volume mesh was created with the commercial mesh generator TGrid. A non-conformal Hexcore mesh (Hexaeder and Tetraeder) was generated with 7 prism layers on the exterior surface of the vehicle and the wind tunnel ground. Additionally the mesh is gradually refined near the vehicle surface and the wake area as depicted in Figure 3 resulting in a total number of 95 million cells in the entire computational domain.

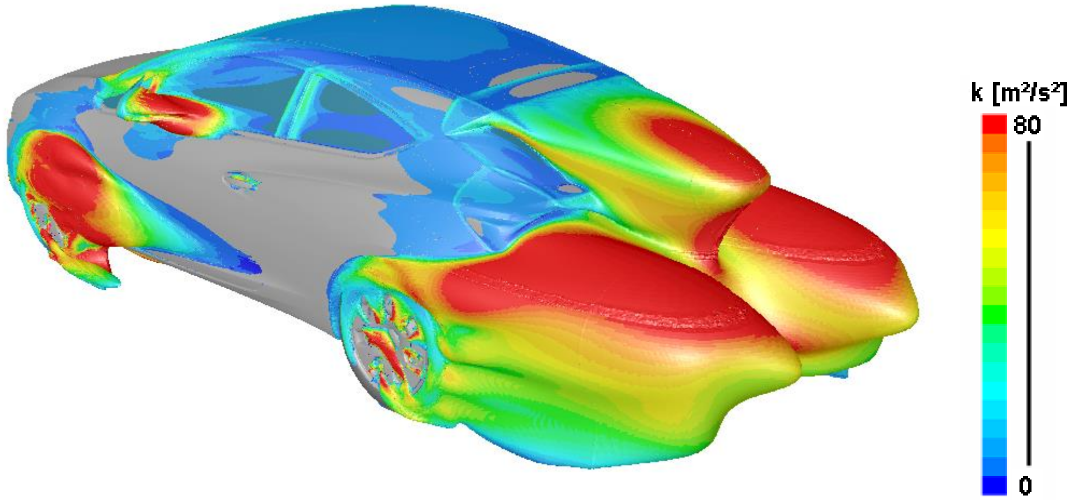

Figure 6: Simulation results *(RANS)* of the complete vehicle model. 3D isosurface of the mean total pressure $<p> = 0$ *bar.*

All boundary surfaces of the virtual wind tunnel (INLET, OUTLET, SIDE1, SIDE2, GROUND and TOP) are identified in Figure 2. At the INLET the velocity was set to a constant value of 140 kph. At the OUTLET a pressure outlet condition is applied. The symmetry condition is used on the TOP as well as on SIDE1 and SIDE2 surfaces. In order to approximate the rotation of the wheels at steady state the Multiple Reference Frame Model (MRF) was used. At the GROUND surface a moving wall condition was applied. All other surfaces of the model are set to the non-slip condition using the non-equilibrium wall function since the first cell was placed in the log-layer obtaining typical values of $y^+ \approx 30$ (see Figure 4).

3.2 Numerical results

The flow field around the vehicle is illustrated in Figure 5. The flow is characterized by a big stagnation point at the front part of the vehicle and a smaller one at the windshield. Low velocity values are depicted in the airflow through the engine compartment and underbody as well at the wake of the vehicle as shown in Figure 5a.

The location of the main turbulent structures are identified by the isosurface of the total mean pressure $<p>=0\ bar$ as illustrated in Figure 6. The isosurface is colored by the Turbulent Kinetic Energy k and shows that the main structures originate from each wheel (4 Structures), each sidemirror (2 Structures) and from the rear end of the vehicle (1 Structure).

4. Submodel

In order to simulate the three dimensional and unsteady turbulent characteristics of one of the 7 turbulent structures depicted in Figure 6 a submodel is defined. The submodel definition encloses the rear upper part of the vehicle geometry as shown in Figure 7a.

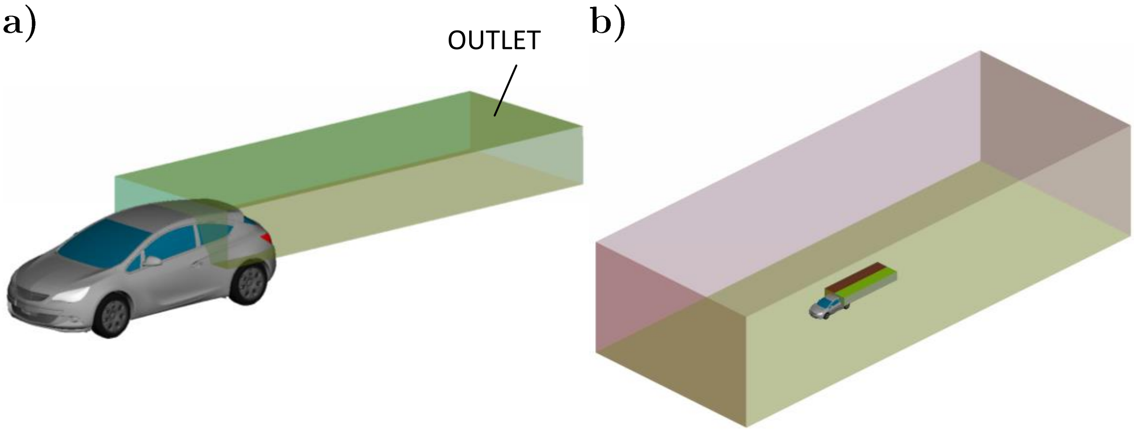

Figure 7: Computation domain of the submodel. a) Boundaries of the submodel`s domain colored as translucent green surfaces. b) Comparison size of the submodel with the wind tunnel dimensions.

In the mainflow direction the subdomain is beginning near the B-pillar of the vehicle, while perpendicularly to the ground the domain is beginning just below the tail lamps. The submodel domain is 7m long, 2.6m wide and 0.15m high. A comparison of the domain size of the complete vehicle model and the submodel is depicted in Figure 7b.

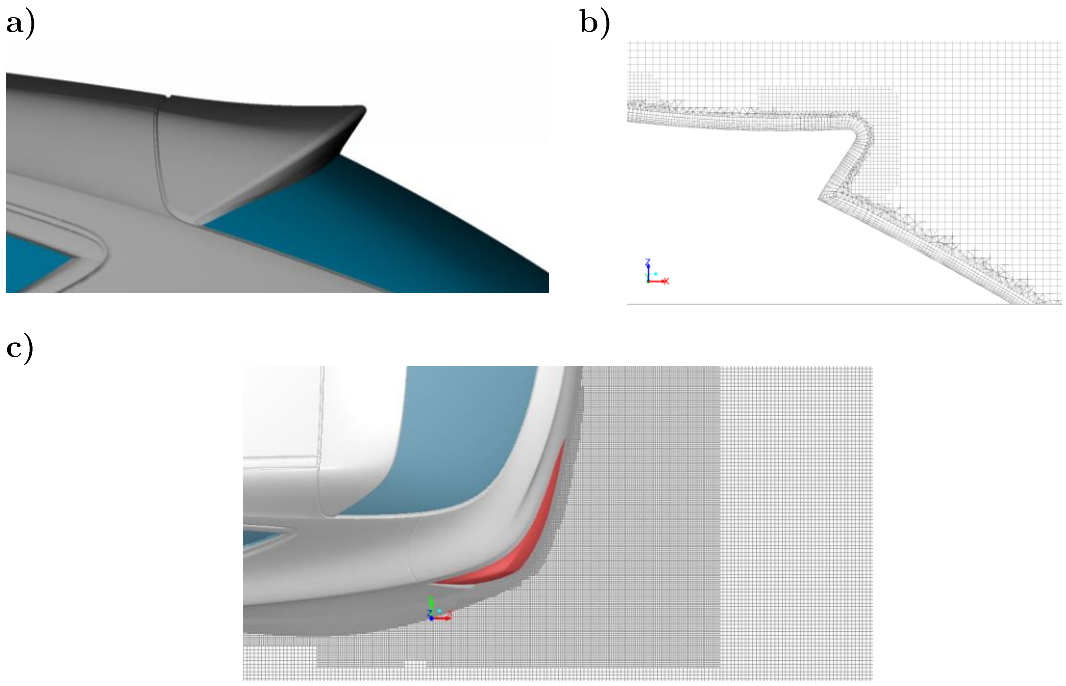

Figure 8: Geometry and computational domain of the submodel. a) Lateral view of the rear roof spoiler of the vehicle. b) X-Z plane of the numerical mesh near the roof spoiler region at the middle of the vehicle. c) X-Y plane of the numerical mesh near the rear part of the vehicle at the height of the tail lamps.

4.1Numerical method

For the submodel a transient simulation was performed using the DES model. The numerical mesh (Figure 8b, 8c) used for the subregion (Figure 7a) is extracted from the mesh used for the complete vehicle model (Figure 2).

A pressure outlet condition was applied at the OUTLET surface. At all others boundary surfaces of the subdomain all three mean velocity components and the turbulence variables k and ε are retrieved from the full vehicle model and given explicitly as inlet conditions.

4.2Numerical results

The non-stationary movement of the turbulent structures created at the rear upper part of the vehicle was captured by isosurfaces of the vorticity ω=1400

1/s as shown in Figure 9a and 9b. In figure 9a three different turbulent structures are identified.

The origin of the first one is the central part of the rear roof spoiler. This structure is elongated parallel to the border of the spoiler and is transported through the flow to regions of higher velocity magnitude. Further upstream this big structure is breaking up into smaller ones as depicted in Figure 9b.

The second structure originates in the corner of the rear roof spoiler, while the third one originates in the slightly curved area between the rear window and the tail lamp. Both structures are aligned with the main flow direction and especially the third one is elongated in the same direction.

Comparing Figure 9a and 9b it is visible that the frequency of the movement of the three structures is different. The structure 2 and structure 3 exhibit a higher level of vorticity as recognizable in Figure 9c.

5. The need of exascale capabilities

The deep understanding of the external aerodynamics of a vehicle is a key topic for a successful automotive design since it influences the vehicle efficiency and the ride stability. The airflow over bluff bodies in the vehicle like the sidemirror, wheels or the rear roof spoiler are characterized through three dimensionality and an unsteady turbulent flow field.

A Reynolds Averaged Navier-Stokes (RANS) approach is able to predict important engineering quantities as the mean flow field, the lift and drag coefficients. However RANS methods have difficulties with the accurate prediction of flow phenomena found around bluff bodies like vortex shedding, flow separation, reattachment and recirculation zones since the influence of the vortices are completely modeled. In the Detached Eddy Simulation (DES) approach, the unsteady RANS models are applied in the boundary layer in order to model the turbulent effects, while the Large Eddy Simulation (LES) treatment is employed to the separated regions in order to resolve the fully three-dimensional turbulent movement of the big eddies.

Simulating the turbulent flow increases the accuracy of the flow prediction, however the CPU requirements of its calculation also increase. The total run time of the RANS calculation of the complete vehicle model (95 million mesh cells) using 95 cpus was 24 hours. For the DES simulation (timestep=0.0001s, totaltime=0.4s) of the rear vehicle submodel (29 million mesh cells) the total runtine was 8 days with the use of 192 cpus.

In order to have fast turnaround times (<36 hours) using the numerical approaches that completely resolve the turbulent scales of a submodel domain or a complete vehicle model domain the exascale capabilities are required. This needs to be demonstrated with an ExaFLOW code.

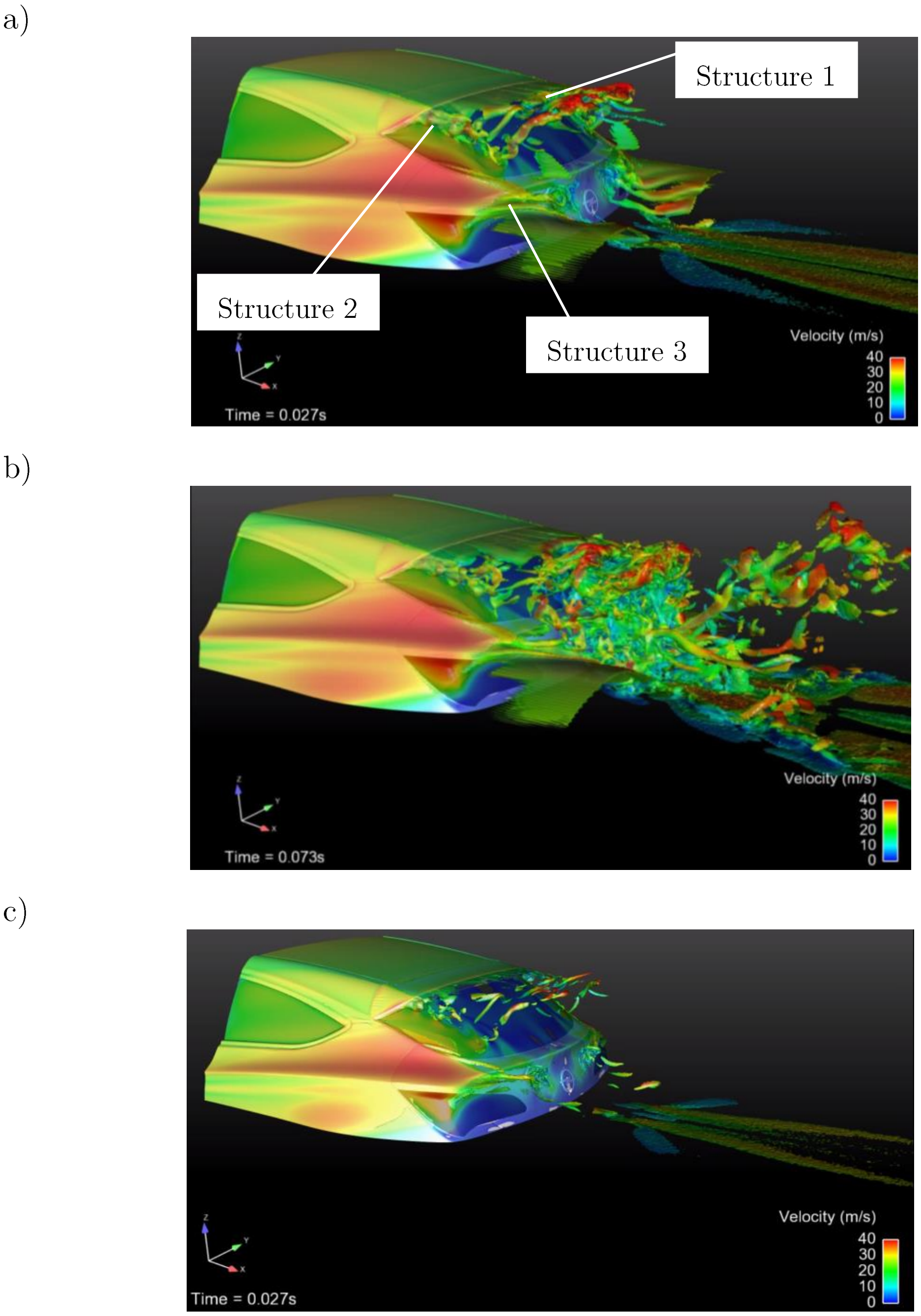

Figure 9: Simulation results *(DES)* of the submodel. a) Isosurface of the vorticity (ω=1400 1/s) at t=0.027s colored by the velocity magnitude. b) Isosurface of the vorticity (ω=1400 1/s) at t=0.073s. c) Isosurface of the vorticity (ω=2300 1/s) at t=0.027s.