

# CFD Tutorial: CAD Cleaning, Solution Computation, and Results Generation.

– Assignment 1 2023

Abhishek Dhiman, (abhdh352)  
Tarun Teja, (tarna588)

# 1 Introduction

The report is a tutorial guide for CAD clean and surface mesh generation (ANSA v20.1.3), volume mesh generation (Ansys Fluent 2020R2), and post-processing (ParaVsview 5.4.1). The significant development in CFD has resulted in the enhancement of the capabilities to perform simulations on large-scale complex geometries e.g. flow around aircraft and road vehicles. However, the computational cost and turnaround time are still an associated issue. Hence, various sub-steps are incorporated at various stages like pre-processing, computation, and post-processing resulting in reduced computation time. The aforementioned tools also support seamless automation scripting as well to further enhance deliverable time. ANSA is a powerful industrial scale pre and post-processing tool for advanced Computational fluid dynamics (CFD) simulations which is used in this tutorial for CAD cleaning and surface mesh generation. The generation of volume mesh and automation scripting for the same is executed in Fluent which is explained in detail in the following sections. Finally, the post-processing of the simulation data to generate pictorial representations on the required results, in this tutorial this step is performed in ParaView and an automation script (python) is generated for the same.

This tutorial describes the details of the steps involved in CFD analysis and the automation scripting associated with a few stages to speed up the deliverable time. The study aims to simulate the flow around a road vehicle (car) moving on the road at a speed of 54 km/hr (15 m/s) in normal weather conditions, perform mesh verification, and automate the entire end-to-end process. The learning gains are the approach for conducting CFD analysis of an industrial or academic nature, brief exposure to automation scripting, and the automation of generating results for post-processing.

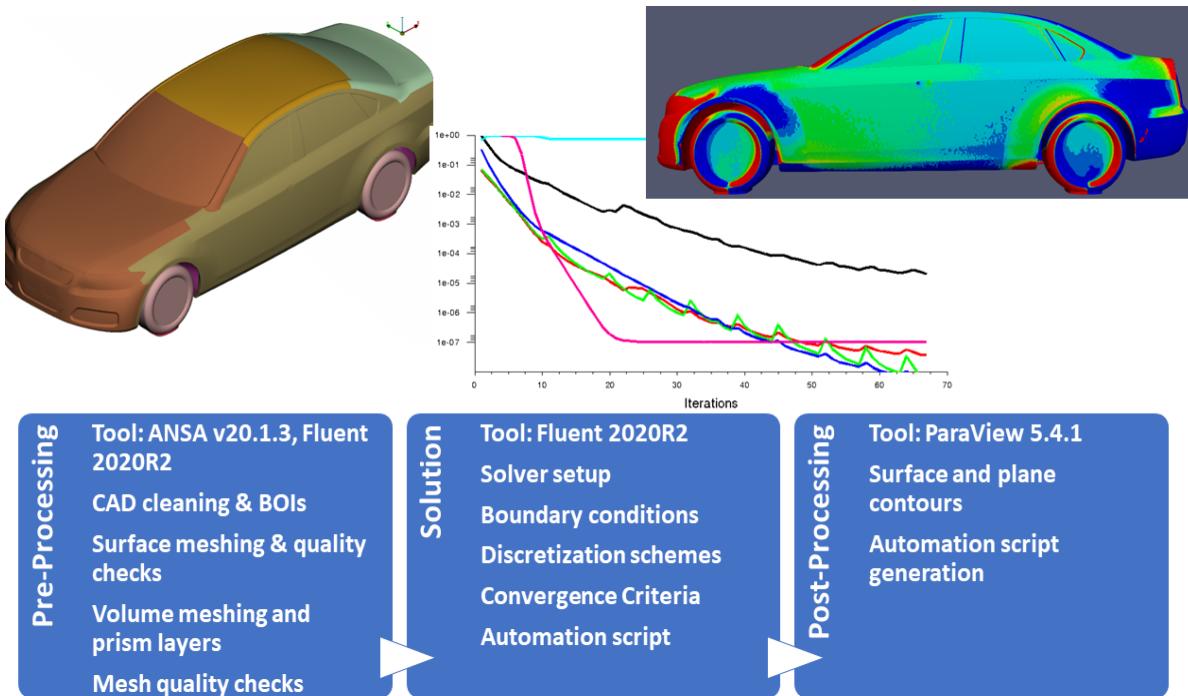


Figure 1: Activity flow chart

# 2 Pre-Processing Description (ANSWER)

The general pre - processing procedure followed in this tutorial is as follows:

- Read the detailed CAD model into ANSA.
- Perform CAD clean i.e. fill small gaps/holes, surface connectivity, removing unnecessary parts/surface from an aerodynamic point of view or least areas of interest.
- Perform geometry checks for intersections, proximities, miss alignments, etc.
- The Next step is the surface mesh generation on the cleaned model, performed mesh quality checks post mesh generation. The off elements must be removed by improving mesh quality.
- For this tutorial, two bodies of influence (BOI) are created in ANSA itself for the regions of interest and capture the flow physics better.
- Another important step is to set the IDs (PID) for the surfaces/regions of the model, domain, etc. These PIDs are important to recognize as the surfaces in fluent solver and in post-processing.
- The Final step in ANSA is to export the surface mesh files into fluent format. The files must be exported separately for the main domain and BOIs.

## 2.1 CAD Cleaning

Launch ANSA and select the CFD option from the popup launcher so that ANSA begins with the default CFD layout. In the toolbar go to File > Open > select the CAD file > press open.

**Note** that when opening a CAD file, ANSA will automatically open the settings window in the Translators section ((Tools > setting) > Translators) thus prompting the user to adjust the settings according to an application of topology and the resolution. For the present study, the hot point matching distance is = 0.05, and CONS matching distance = 0.2, and the tolerance mode = middle. This setting allows for better connections of faces and curves, if tolerance mode = fine or extra fine the resulting topology is poor and more non-connecting faces. The rest of the settings are kept to default. The cleaning is done in the topology part of ANSA where geometric changes are made, the body of influence is created and undesirable CONS (red and cyan) are eliminated. The red CON implies a single free edge and cyan CON for the connection of three or more faces. The yellow CON is desired in the entire model which implies only two connecting faces.

The clean-up activity for the car model includes the following and can be seen in Fig. 2:

- The missing surfaces/holes are filled with various options provided in ANSA to maintain surface continuity.
- The door handles are replaced with the plane surfaces as the aerodynamic obstruction is lesser compared to the entire side surface.
- Similar to door handles, the side view mirrors are also removed. These parts are mandatory in a road vehicle and contribute marginally towards the flow overall flow over the car body.
- Next is the removal of the grills from the front section. These grills are used for engine and radiator cooling which are positioned inside the car (internal flow). The region of interest in this study is the external aerodynamics of the car body hence grills are replaced with the planar surface.

- The alloy wheels or rims are primarily used for weight reduction, corrosion resistance, better disk brake cooling, and as alternatives to heavier steel wheels for the car racing industry [1]. In commercial vehicles, the general use is for aesthetics and lightweight. Hence, the rims are replaced by a planar surface as high-speed braking and disk cooling are not a significant concern for commercial vehicles.
- The sharp curvatures are made smooth and the surfaces with very small areas are combined with adjacent faces to make the surface bigger and better for meshing.
- The road horizontal is not a tangent to the car tyre as due to the weight of the car the tyre gets pressed against the road and makes it a surface contact. This also gives additional benefits in volume meshing.

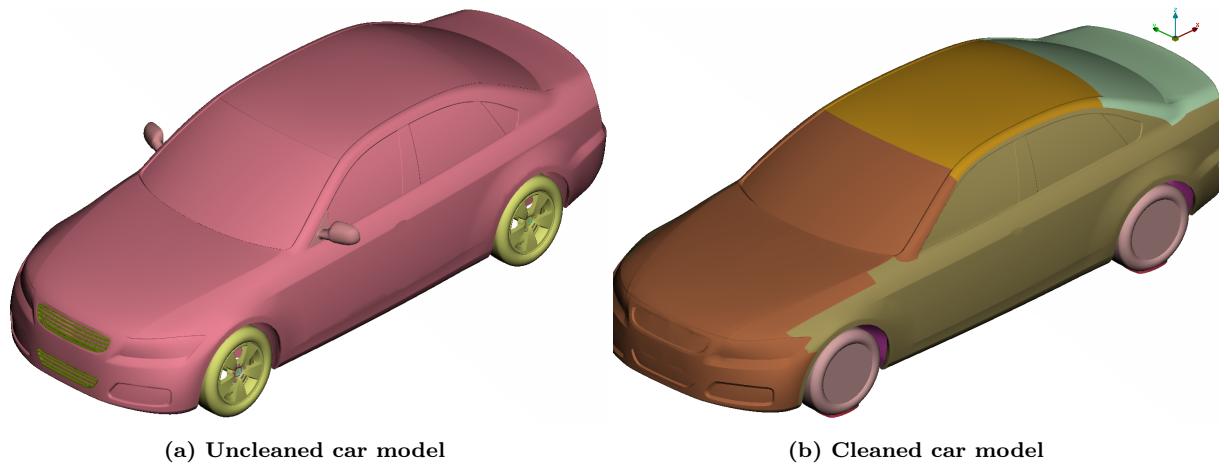


Figure 2: The cleaned car model showing different surfaces i.e. front, top, side, rear, wheels

## 2.2 Domain and body/bodies of influence

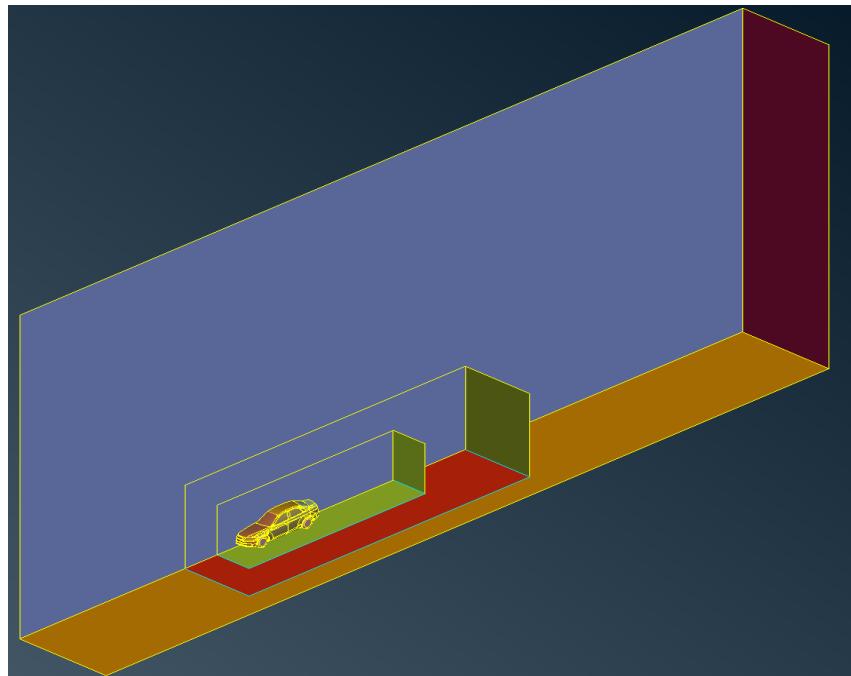


Figure 3: Symmetric domain with inner and outer BOIs for smoother mesh growth

The creation of fluid domain and body of influences around the car body are important before going ahead with surface meshing, Fig 3. The body on influence is used as regions of mesh refinement to capture the salient features of the flow in the region of interest i.e. car model. The characteristics length ( $L$ ) is taken as the longitudinal length of the car and the center plane dividing the car into two equal halves is used as the symmetry plane. The symmetry plane is used to divide the entire domain and car body into half such that the size of the volume mesh can be reduced to save computation time and deliverable time. This is explained in detail under the boundary conditions section. The important observation in Fig. 3 is the domain size with origin at the car front. The upstream boundary is at  $3L$  from the origin to have developed flow,  $7L$  downstream to avoid back pressure,  $4.3L$  for the top, and  $1.2L$  on the car's left side.

## 2.3 ANSA: Surface Meshing

A good quality surface mesh is important as it serves as the base for a good volume mesh. The uniformity of the element size is also to be maintained while doing surface mesh. A useful tip is to set the PIDs for all the surfaces and boundaries before starting the surface mesh as it will ease the activation/deactivation of the surfaces for surface meshing. Now, coming to ANSA mesh mode, the perimeters section allows to apply sizing on the edges (perimeter) and surfaces (macros). Goto Perimeters > Spacing > Auto CFD > perimeters/macros and then set the minimum and maximum element sizes for the selected edges/surfaces. The above step only defines the sizing, to generate the mesh for the same goto Mesh Generation section. There are two techniques that are used for this tutorial i.e. Adv. Front and CFD, where first is an advanced algorithm that is used to get uniform mesh on the entire surface under the pre-defined sizing and later (CFD) is used to profound variable size based on the local curvature and sizing. The triangular mesh elements are selected for the surface mesh instead of the tri/quad combination or quad dominant as it captures the complex geometric intersections of the surfaces well [2]. Aids in the generation of prism layers and uniformity of element distribution during volume meshing in Ansys.

The disadvantage of Adv.Front is the large number of elements compared to the CFD algorithm. Hence, for surfaces with high curvatures Adv.Front is used like wheels, ground-wheel contact, front and rear of the car. Whereas, CFD is used for more flat and large surfaces like car top, side, and bottom. Show only one PID at a time to mesh it, if satisfied with the element distribution then check the mesh quality for off elements. The mesh quality is based on the Fluent criteria standards for element skewness (0.5), and  $\min(30^\circ) - \max(1.7*60^\circ)$  angle for tria and quad elements. In Fluent maximum skewness recommended is 0.95 i.e. 95% and the angles to be closer to  $60^\circ$  [2]. ANSA gives another useful function under Shell Mesh i.e. improve, which refines the edges and nodes quality to resolve off elements. All the off elements must be resolved, once done then freeze the mesh and activate the next PID. Freeze/Unfreeze is another useful function under Macros which basically mesh lock / unlock. Once a PID mesh is frozen the mesh cannot be altered and the associated surfaces can be activated and meshed. This process continues for all the PIDs one by one which results in the surface mesh on all the surfaces. The surfaces are created with the default conformal mesh which results in pyramid elements in the volume mesh at the intersection. A non-conformal approach (interface elements not aligned) is also supported in ANSA which will prevent pyramid elements in volume mesh and is more robust [3]. However, the flux transfer across the non-conformal element faces is calculated between the overlapping zones by creating additional division on the faces [2].

Next is the export step post surface mesh creation on the car body, domain, and BOIs. The important thing to note is that the BOIs are to be exported separately in a separate file and the car and domain boundaries are in another file. For export goto File > output > fluent a pop comes up make sure to check the file format as binary, un-check scale as it can be set in the fluent solver automation script, and save the file.

## 2.4 Fluent: Volume Meshing

After generation of the surface mesh the next step is to generate the volume mesh which is performed in Fluent with scheme files in order to automate the end-to-end process.

The general layout of the automation scheme file is as follows, the details and the values for certain variables are provided in the later sections:

- 1 Define the variables for PIDs, global and local size settings, and names of the objects that will be created in fluent mesh mode during the process.
- 2 Read the Fluent mesh file for the domain and BOIs in the script file. Note, that the mesh files should be in the legacy format (.msh), else an error will be displayed.
- 3 Once the mesh files are imported, the PIDs will be read as un-referenced objects in Fluent meshing mode. Hence, the geometry (goem) object is created for BOIs as these are just boxes for the refinement zone, and the mesh object is created for the fluid domain. Post this step all the un-referenced zones will be cleared.
- 4 Next is the setup of element sizing which is divided into global sizing (entire fluid domain) and local sizing (each surface / PIDs). The global and local sizing is based on the size of the surface mesh in order to achieve smooth element growth and avoid abrupt changes in cell aspect ratio. Hence different local sizing is applied for the car body, wheel-ground contact, and domain surfaces to have finer mesh around the car model and coarser as it moves towards the boundaries.
- 5 The sizing fields on BOIs are based on the level of refinement needed and should comply with the surface mesh created for these on the domain faces. The Mesh verification is performed for coarse medium and fine mesh, hence different sizing are set for each. After setting up sizing fields, perform validate and compute operations.
- 6 Now material point is created inside the domain as a pointer/reference point for the fluid domain. It can be created anywhere inside the domain but away from the car model and boundaries. Post this point creation, compute the volumetric region based on this point resulting in a new under under the volumetric region which is the fluid domain.
- 7 Next is the sizing of prism layers (inflation layers) for the car surface and the ground to keep the elements transition smooth and avoid termination of prism layers at the wheel-ground contact. Set the growth option to ICEMCFD-quality with a default value of 0.98 and have sufficient layers (at least 11) in the present tutorial 12 prism layer is created to capture the boundary layer on the car surface. The k -  $\varepsilon$  realizable (RKE) turbulence model is used with enhanced wall treatment to resolve the boundary layer for the fine prism layer at a high Reynolds Number ( $R_e$ ). Also, at high  $R_e$ , RKE is more stable with the boundary conditions, unlike SST.
- 8 Next is to fill the core domain with elements. For this, hexcore elements are selected because the total number of elements reduces significantly compared to tetrahedral elements. It has more faces per cell which improves information sharing between the cells and the convergence is faster than tetrahedral. Another better option is poly-core mesh or a hybrid of poly-hexcore which reduces the overall cell count even more resulting in a coarser mesh. The poly and poly-hexcore meshes are not used in this tutorial.

- 9 Coming towards the meshing part now, use the Auto-Mesh functionality to generate the volume mesh. The pyramid elements are used for the transition from prism layers to the hexcore elements. The merge cell zones option must be checked at this point as there will be two fluid zones created prism and hexcore which must be combined to get one single domain.
- 10 The auto-mesh takes time to generate the volume mesh, after which the mesh quality improvement is performed. The ICEMCFD quality method is used for improvement which results in a good quality volume mesh. The quality = 1 - Ansys ICEM CFD quality (0.98), where 1 is the ideal element quality implies that highly distorted elements that are susceptible to cause convergence instabilities are improved [2].
- 11 Check the mesh quality and cell counts and perform the "prepare to solve" step which will delete the dead zones, edges, geometry objects, unused faces, and nodes. Basically, a final clean-up before the solution.
- 12 The generated mesh can be seen in Fig. 4, where the volume and BOIs refine are captured along with the prism layer distribution near the wheel-ground vicinity Fig. 4b.
- 13 Last step is to export the mesh file into legacy mesh format such that it can be used as an automation scheme file for the solver.

Note The mesh generation automation scripts are attached along with this report for reference. The automation scripts run the Ansys TUI command which speeds up the entire process.

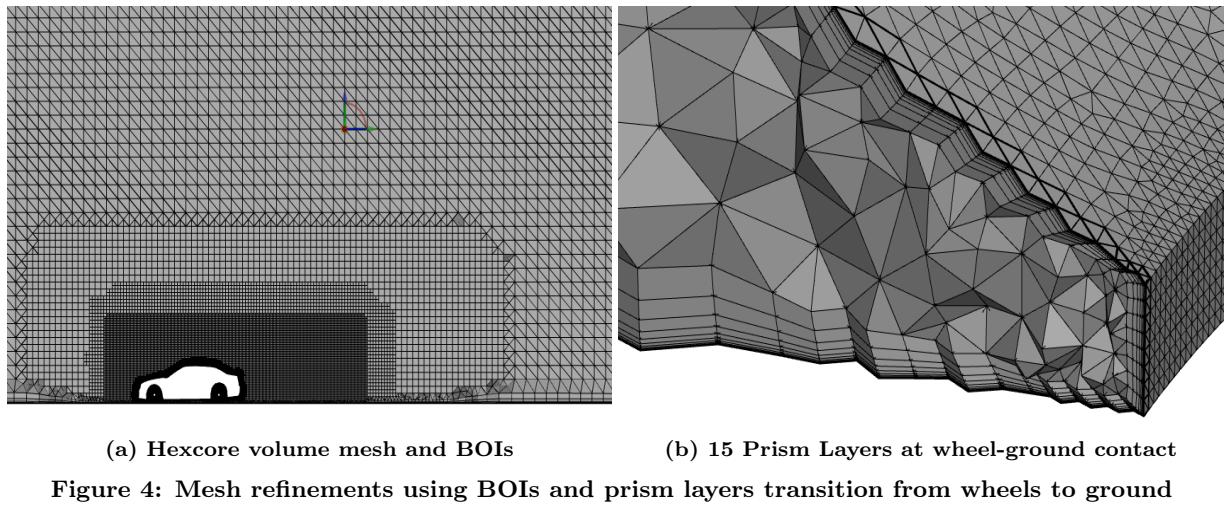


Figure 4: Mesh refinements using BOIs and prism layers transition from wheels to ground

## 3 Solution

### 3.1 Solver setup and boundary conditions

The case of a car moving at a speed of 54 km/hr (15 m/s) is taken as the speed of 50-60 km/hr is the average speed for a car on city roads and for normal travel. The car length is considered as the characteristic length ( $L = 4.625$  m) resulting in  $Re_L = 4.75e^{-6}$  [4]. For the simplicity of the problem, weather conditions are selected such that there are no crosswinds, wind speed relative to the car at average speed is negligible and the car is moving on a straight road hence experiencing straight flow from the front. The domain size is mentioned in the Sec. 3.

The selected boundary conditions are as follows:

- Velocity inlet is set at domain inlet face with velocity components where only x-component is initialized with 15 m/s and the rest two are 0. Therefore establishing unidirectional flow and no cross winds. The Atmospheric Boundary Layer (ABL) is a good consideration leading to a velocity profile at the inlet and is popular for Urban Flow Physics to account for small objects in surface roughness. However, in this particular case, the focus is on a single object moving at high velocity resulting in a more uniform profile.
- Pressure outlet is set at domain outlet face with gauge pressure = 0 implying atmospheric pressure.
- The top, left and right side domain faces are initialized with symmetry conditions as no crosswinds are present and flow will be symmetric if the longitudinal symmetric car body is taken. This cuts the domain in half creating room for mesh refinement around the car body and reduction in computation time.
- The car body surfaces are set to the wall with no-slip condition. Whereas, the ground is set to a moving wall with no-slip conditions where the moving wall velocity same as the upstream flow velocity. This means that the car is not stationary, instead moving with a velocity of 15 m/s.

The solution is initialized with a steady-state and pressure-based solver as the flow speed is low (0.045 M 0.3 M) thereby giving negligible density changes, where the speed of sound is taken at sea level on a standard day. The choice of turbulence model is  $k - \epsilon$  Realizable with enhanced wall treatment due to its capabilities of wall modeling as per the mesh refinement near the wall and the stability with the boundary conditions at high  $Re$ . The model is suited for adverse pressure gradients which are strongly associated with the flow physics in this case. The coupled solver is used for pressure-velocity coupling which solves both momentum and pressure-based continuity equations together. For spatial discretization of pressure and momentum, the 2<sup>nd</sup> scheme is selected. Whereas, the turbulent kinetic energy (TKE) and specific dissipation rate are initialized with 1<sup>st</sup> upwind. The gradient is set to least square cell based which gives better performance on irregular surfaces corresponding to car-body curvatures and is computationally inexpensive. The domain is initialized with the hybrid scheme which solves Laplace's equation to obtain velocity and pressure field in the entire domain based on the boundary condition. The time scale factor is also important to achieve smooth convergence, initially set to 1 which is sufficient as per the boundary conditions. But whenever divergence is observed in the residuals, it is reduced in the step of 0.1 and the lowest value of 0.7 was set to achieve good convergence, Fig. 5

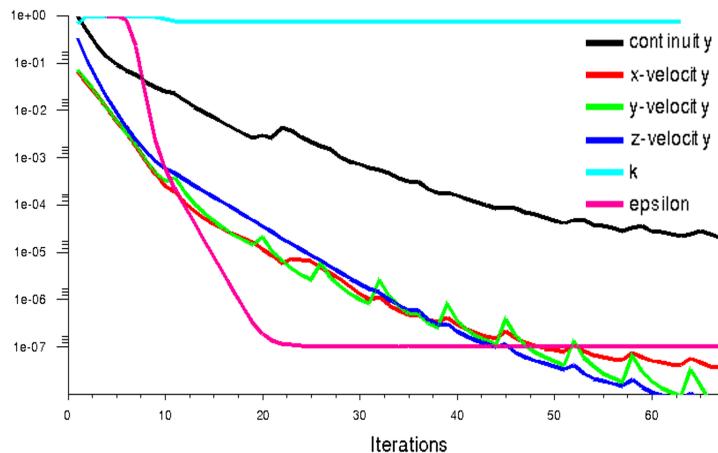


Figure 5: Residual convergence achieved for the study

All the above-mentioned settings have their respective TUI commands which are used in the scheme file for the solver in regards to the automation [2]. Additionally, the case and data files, lift, drag, and other variable values are exported into separate text files for each iteration in order to be used in the post-processing stage. The solver scheme file is attached for reference.

## 3.2 Mesh Verification

The reliability of the study is very important to convince the reader that the methodology followed is suitable for the corresponding case study and that the findings are trustworthy. Hence, a mesh verification study is performed in order to quantify discretization errors and identify a suitable grid. The methodology proposed by Celik [5] is widely accepted in the CFD community, recommending a grid refinement factor  $> 1.33$  but dependant on the industry e.g. automotive  $> 1.44$ . The x and y forces/coefficients on the intended body are monitored along with the residuals. The Grid Convergence Index (GCI) is then calculated as by using the procedure recommended for RANS in [5]. A suitable mesh with a low GCI (preferably  $< 5\%$ ) and no. element is then opted for further analysis and results. For Scale Resolving Solvers (SRS), the mesh verification is conducted differently as mentioned in [6] [7]. The current is limited to RANS models.

# 4 Para-View Automation Steps

## 4.1 Intro to ParaView

ParaView is an open-source software (OSS) for quantitative visualization and is widely used for post-processing work. The amount of data sets that can be handled by ParaView depends on the available computation power e.g. small-size data for daily use laptops and huge data sets for supercomputers. The ParaView-supported platforms range from single-processor workstations to multiple-processor distributed memory supercomputers.

Data is the new oil, especially when it comes to the aerospace and automotive industries which generate huge amounts of CFD data in order to reduce the massive bills associated with experimental tests. Hence, the visualization of this data becomes significantly important in identifying complex structures, parameter variations, conducting benchmark studies, etc. Better visualization enhances the comprehension of simulation results. Hence, new and powerful tools like ParaView are developed to cater to the industry's requirements.

There are three basic steps to visualize the data: reading raw data, filtering/cleaning, and generating appropriate plots/contours/figures best suited for the intended reader. In terms of the Fluent to ParaView transition, the generated data file (encase-gold format i.e. encase) is read into ParaView which internally calls the associated data files exported from Fluent. After importing required data multiple filters can be applied to generate, extract, or derive the features from the data. Finally, a viewable image is rendered from the data and exported out of ParaView.

## 4.2 Para-View Automation

The results obtained in this study are limited to the Pressure Coefficient ( $C_p$ ) distribution on the car body (6, 8 and 7) and Total Pressure Coefficient ( $C_{p_{tot}}$ ) on longitudinal (Fig. 11) and lateral (12) planes along the length and width of the car respectively. Also, the distribution of  $C_p = 0$  and  $C_{p_{tot}} = 0$  in the domain and around the car body, Fig. 9 10. The  $C_p$  distribution quantifies

the push and pull forces on the regions of the car body, and  $C_{p_{tot}} = 0$  establish the extent and regions where the pressure of normalizing to be equal to freestream.

- To generate the automation script for post-processing, let us get started using ParaView. As ParaView is an open-source software it can be installed on a normal laptop/computer. For this tutorial, ParaView version 5.4.1 is utilized. Before loading the data open the .encas in a text editor and remove the double quotes (").
- Click on File menu > open > select the .encas file generated from Fluent. A popup window will appear to select the file format, click on Ensight files and press OK. Note that opening a file is a two-step process, hence you do not see any data yet. Instead, you see that the properties panel is populated with several options including pressure, wall shear components, y-plus, velocity components, and mass imbalance. For the present tutorial, only pressure and velocity are required hence select, then click on the Apply button.
- The Filters menu holds all the operations that can be performed on the data. Select the Extract block filter to extract the Fluid domain to generate the uniform cut section.
- The Next step is to scale down the domain into smaller ones near the car body in all directions by using the Clip filter. The Slice filter is used on the car body in order to capture the flow around the car body on certain planes, then calculate the  $C_{p_{tot}} = P/(0.5 * \rho * (V_{inf}^2)) + ((V_x)^2/(V_{inf}^2))$  [8] with the Calculator filter. The  $C_{p_{tot}}$  evaluates to a user-defined expression on all the points of the filter region/planes. The Contour filter is then used to visualize the  $C_{p_{tot}}$  along the length and width of the car Fig.11, 12
- The total pressure coefficient ( $C_{p_{tot}} = 0$ ) of the region around the car model is shown in Fig. 9 10. It is estimated by using an extract-block filter and selecting all faces of the car model including wheels in the block entities window and clicking Apply, then implementing the same procedure as done in the previous step.
- Finally, the pressure coefficient for the individual surfaces of the body by using ( $C_p = P/(0.5 * \rho * (V_{inf}^2))$ ) formula in the calculator filter. The remaining procedure is the same as when estimating the total pressure coefficient. The pressure coefficient ( $C_p$ ) from various car views can be seen in Fig.[6, 8, 7].
- After completion of all steps, save the state file from ParaView, generate the required images of the contours/streamlines, etc. This same procedure is followed to generate the automation script (python) manually which generates the results and the images. Another useful option provided in ParaView is to utilize the Trace functionality which records the manual steps and automatically creates the Python script which becomes the deliverable along with the data files. The recommendation is to try these approaches and opt for a suitable one as per individual.

# 5 Results

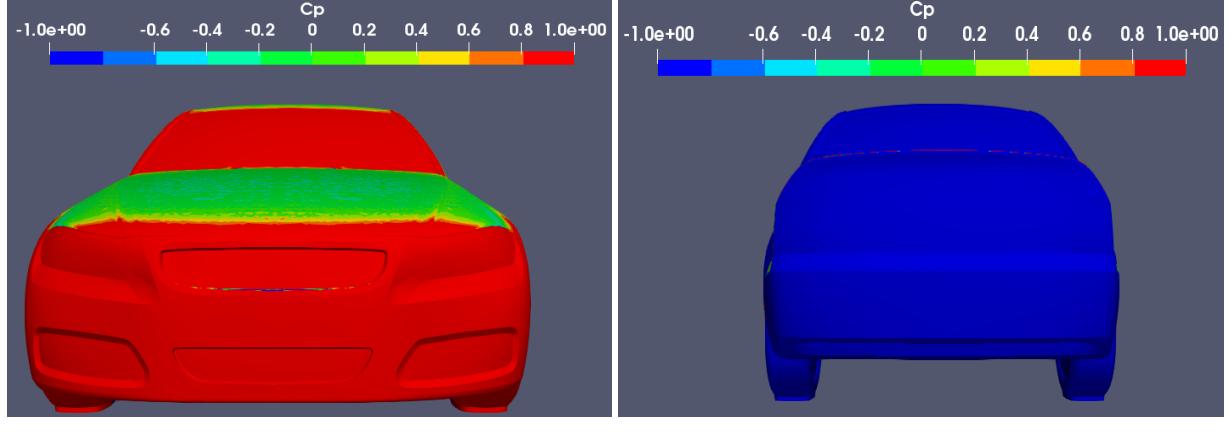


Figure 6: Pressure Coefficient ( $C_p$ ) contours on the front (push force [+]) and rear (pull force [-])

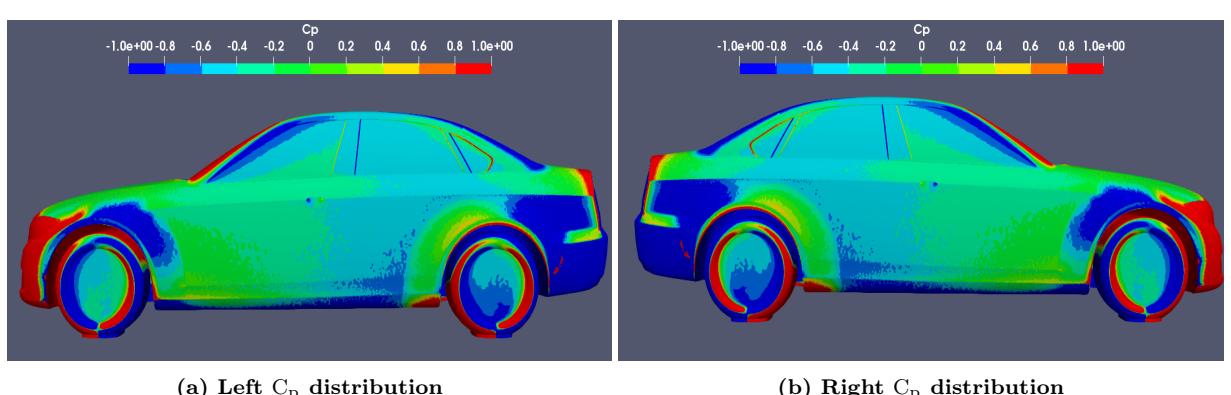
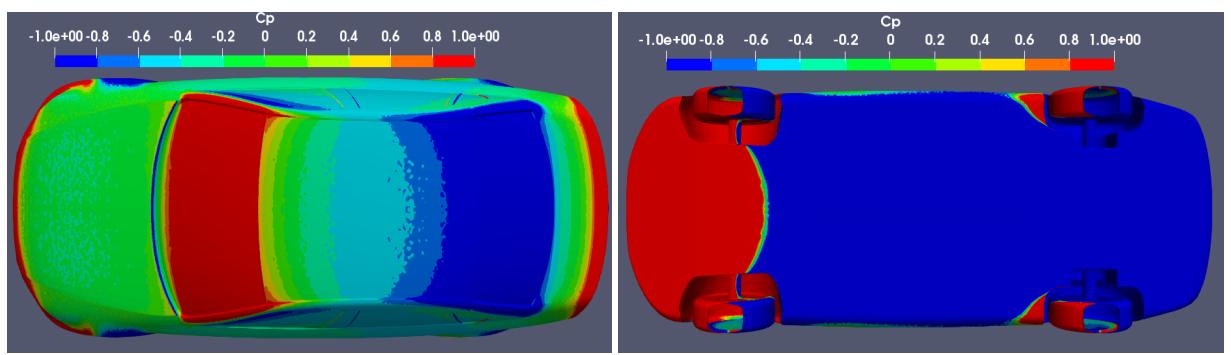
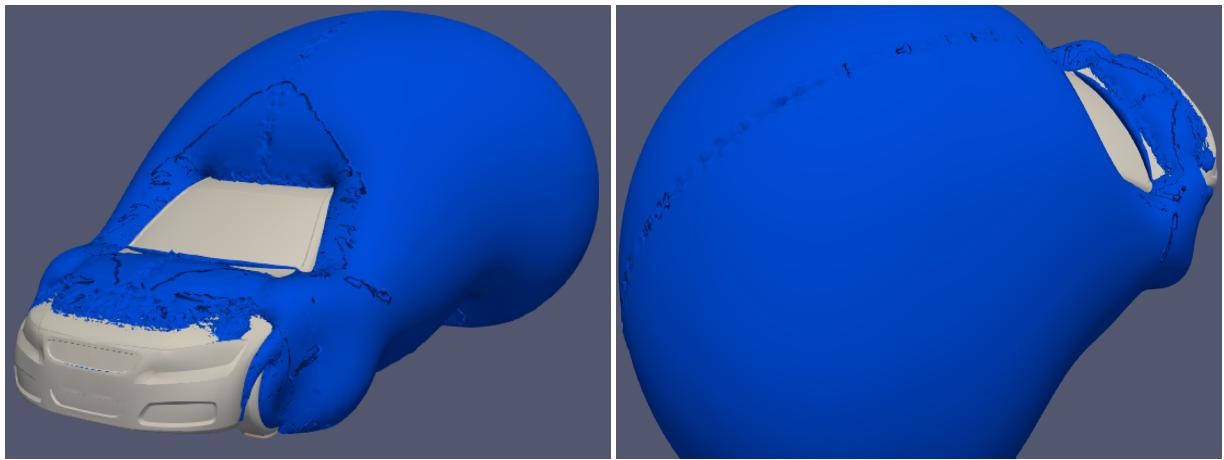


Figure 7: Pressure Coefficient ( $C_p$ ) contours on the left and right sides, comparatively uniform

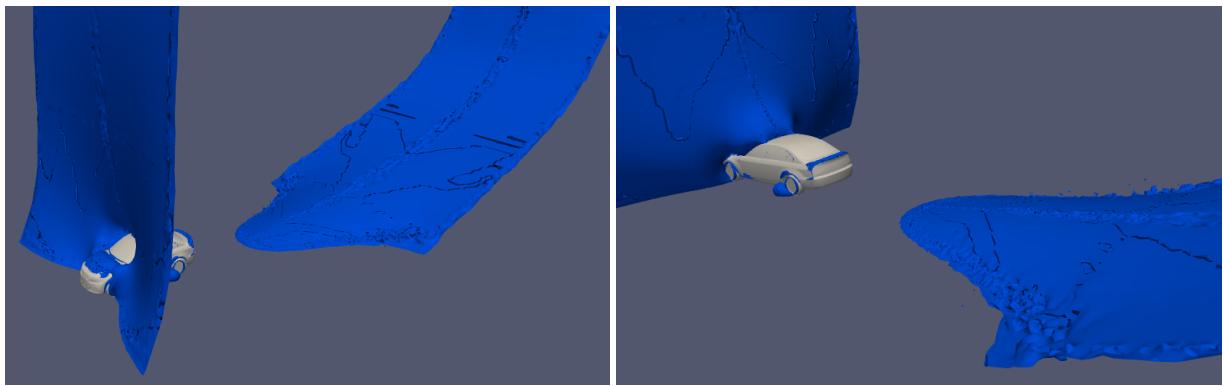


(a) Top  $C_p$ , up force on front, downforce on rear    (b) Bottom  $C_p$ , up force on front, downforce on rear

Figure 8: Pressure Coefficient ( $C_p$ ) contours on the top and bottom



**Figure 9:** Total Pressure Coefficient ( $C_{P_{tot}}$ ) iso-surfaces in the domain



**Figure 10:** Pressure Coefficient ( $C_{P_{tot}}$ ) iso-surfaces in the domain

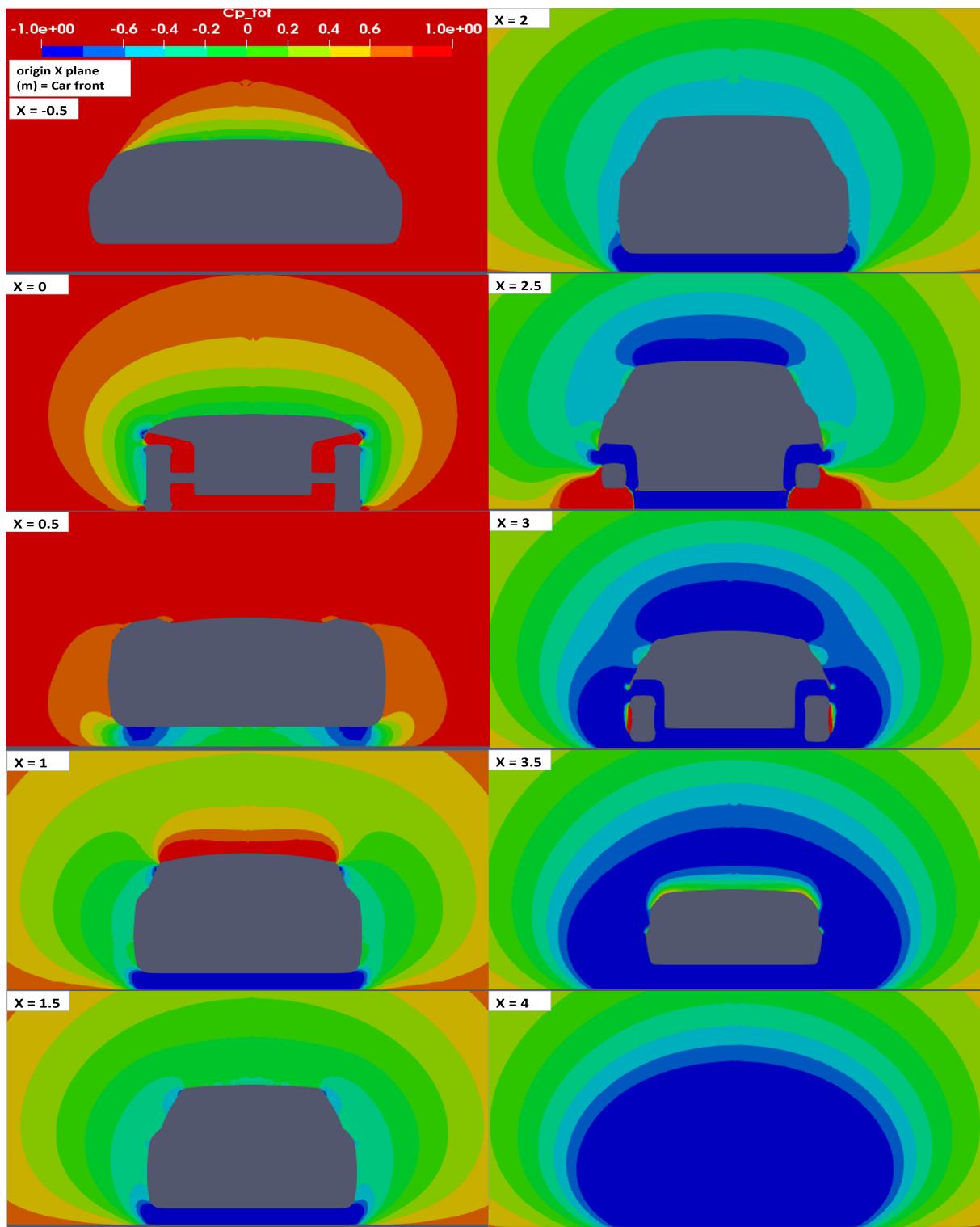


Figure 11:  $C_{p_{tot}}$  distribution around the car body in longitudinal plane

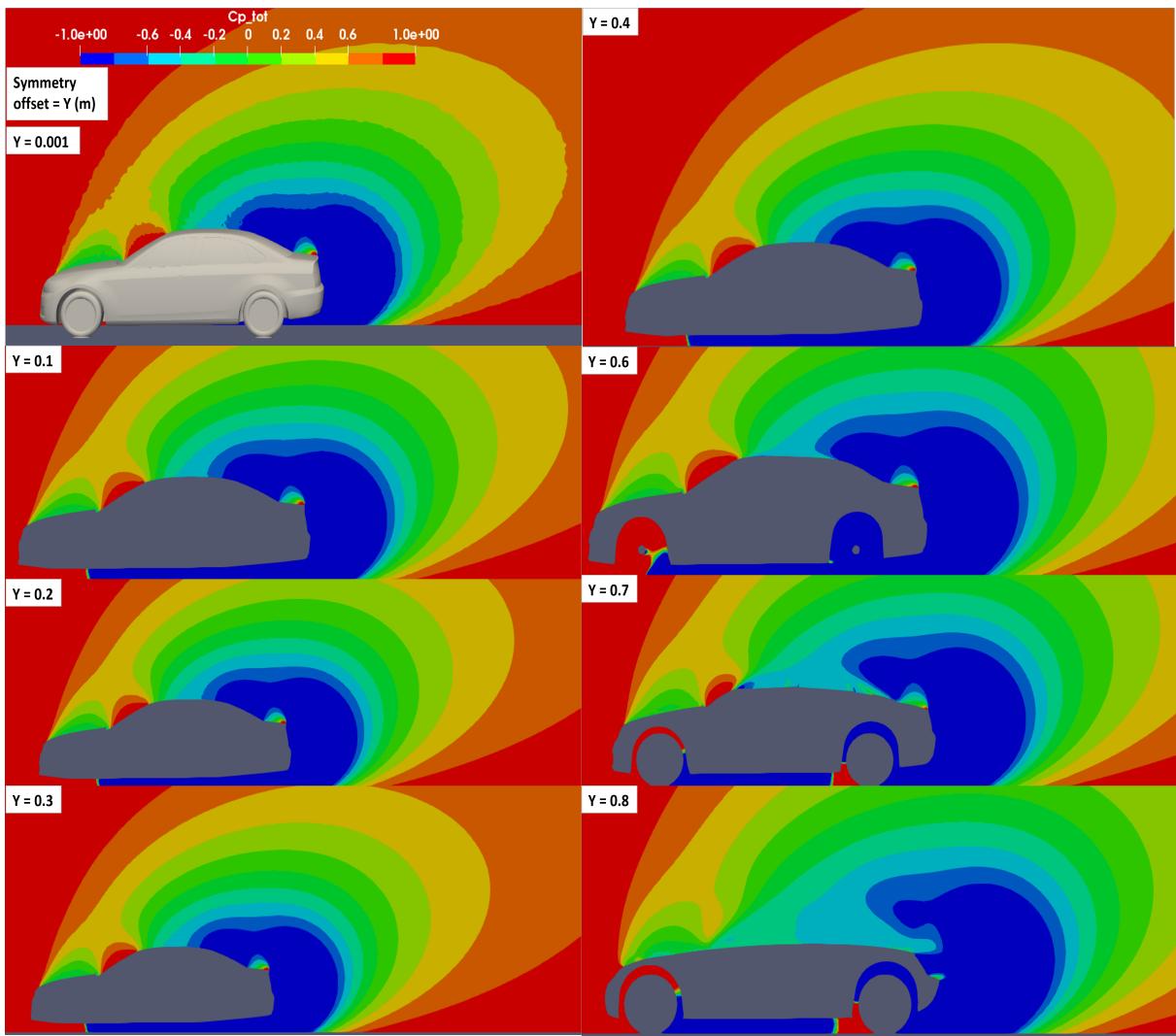


Figure 12:  $C_{p_{tot}}$  distribution around the car body in lateral plane

# References

- [1] WHAT ARE ALLOY WHEELS AND WHAT ADVANTAGES DO THEY HAVE?;. Available from: <https://www.hpdwheels.com/blogs/news/alloy-wheels>.
- [2] Ansys. Ansys Fluent Theory Guide. 2020.
- [3] Systems BC. ANSA version 20 1 .x User Guide. 2019.
- [4] Max Varney FW Martin Passmore, Kuthada T. Experimental Data for the Validation of Numerical Methods: DrivAer Model. 2020.
- [5] et al IBC. Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications. 2008.
- [6] Menter FR. Best Practice: Scale-Resolving Simulations in ANSYS CFD. 2015.
- [7] Johan Meyers PS Bernard J Geurts. Quality and Reliability of Large-Eddy Simulations. 12th ed. ERCOFTAC SERIES, SPRINGER, ISBN: 978-1-4020-8577-2; 2008.
- [8] YAZDANI R. Steady and Unsteady Numerical Analysis of the DrivAer Model. 2015.