A 3D Voxel-based Approach for Fast Aerodynamic Analyses in Conceptual Design Phases

Antonio Bacciaglia[1], Alessandro Ceruti[2] and Alfredo Liverani[3]

[1]Department of Industrial Engineering – DIN, University of Bologna, Italy
antonio.bacciaglia2@unibo.it
[2] Department of Industrial Engineering – DIN, University of Bologna, Italy
alessandro.ceruti@unibo.it
[3] Department of Industrial Engineering – DIN, University of Bologna, Italy
alfredo.liverani@unibo.it

Corresponding author: Antonio Bacciaglia, antonio.bacciaglia2@unibo.it

Abstract. The panel method is a potential-flow numerical approach that shows valuable performances to solve aerodynamic problems in the preliminary design stages. It shows a lower computational effort compared with Computational Fluid Dynamics, wind tunnel tests or 'on the field' experiments. However, the 3D surface discretization in rectangular panels is tedious and must be often carried out manually from scratch. Moreover, the panel method can't be used to compute the overall drag force due to strong assumptions. To solve these two challenging aspects, the authors propose a voxel-based fluid dynamic approach integrating its programmed functions within a panel method. Voxelization is used to automatically distribute coherently the panels along the external surface of a 3D model in an automated way. A parametric study is included to demonstrate how the voxel resolution affects the aerodynamic results and provide guidelines for future research. Overall drag is estimated using corrections for both the skin friction and the form drag sources. The Ahmed body case study is included and demonstrates a good agreement between the voxel-based fluid dynamics approach and the literature benchmarking values, but with lower computational efforts. Further studies involving more complex shapes should be performed to better understand the performances and limitations of the approach.

Keywords: Voxelization, Conceptual Design, Ahmed body, Panel method, CAD, Automotive.
DOI: https://doi.org/10.14733/cadaps.2022.1236-1254

1 INTRODUCTION

One of the main problems that arise in industrial engineering when an object (e.g. cars, aircraft, trains, but also skyscrapers in civil engineering) is surrounded by airflow is the determination of the aerodynamic forces and moments acting on the body itself. The designers can choose different approaches to evaluate the aerodynamic loads. The most precise way consists of the testing of the real-life component, which implies the manufacturing of a new prototype each time a change is introduced. Moreover, full-scale tests require the installation of complex measurement equipment to collect data. As a consequence, the experimental approach is the most expensive both in terms of time to market and money and could be unsuitable in the conceptual and preliminary design stages where a significant number of configurations should be tested and evaluated. Another common approach used by aerodynamic engineers is to test a small-scale object in the wind tunnel facility; however, this approach is expensive and slow, especially in the first design stages. In addition, a wind tunnel facility requires a huge capital investment, especially for small companies. Wind tunnel tests require also a long and painstakingly setting up and calibration to obtain consistent results. Nowadays, scaled models can be obtained exploiting the capabilities of Additive Manufacturing, but each time a change is introduced there is the need for manufacturing new parts. For this reason, fluid dynamics numerical approaches are often used to evaluate the velocity and pressure fields around objects in the conceptual design stage.

Among the numerical approaches to compute the aerodynamic loads, the panel method is one of the simplest methodologies to solve the potential flow. It gained popularity during the 1970s especially in the aerospace sector (e.g. Panair code by Boeing) [6], and it is based upon strong flow hypotheses as inviscid, incompressible, irrotational and steady flow [14]. Thus, the resulting flow field of the panel method is exempt from viscous effects that affect drag considerably. For this reason, in the last decades, this numerical approach was ignored, while the attention shifted towards the Computational Fluid Dynamics (CFD) approaches that at the moment are considered the most valuable, accurate, fast and cheap strategy to solve aerodynamic problems. The CFD software is based on a discretized formulation of the Navier Stokes equations that are solved for each finite volume element the flow domain is discretized. Indeed, CFD uses volumetric discretization, while the panel method works with a surface subdivision of the object into simple elements called panels. This discrepancy reflects on the computational time required to find a solution: the CFD approach needs a longer time to solve the problem, even if it is more accurate than the panel method. As a consequence, the panel method is still a valuable methodology to evaluate the aerodynamic loads and moments in the first design stages, when different configurations should be analyzed in short times to define the optimal one that deserves further development. Once the best configuration is found, the CFD approach can be used to precisely estimate the aerodynamic performances of the object: a possible design workflow has been depicted in Figure 1 for reference. Based on this approach, wind tunnel tests could be performed only when the detailed design is fixed to validate the CFD results; finally, experimental tests could be performed on built prototypes to contain as much as possible the R&D costs. Just to give a simple comparison in terms of time needed to set up an aerodynamic test, experiments 'on the field' on a real prototype may take several weeks or even months to build the prototype, set and calibrate the sensors, collect the data and analyze them. On the other hand, tests carried out in wind tunnel facilities may take a similar time duration due to the need for building a scaled version of the object, to set and calibrate the sensors, and analyze the data collected. However, the overall costs may be lower compared to experimental tests on real-life objects. Differently, numerical approaches shorten the aerodynamic analysis to a few hours to design the 3D model, mesh the flow domain, set and run the simulation. As of last, the panel method can even perform faster compared to CFD programs due to the approach's easiness and rough aerodynamic estimation that can be obtained in a few minutes.

Apart from the strong physical assumptions of the panel method, the main criticality of this numerical approach is the 'panelization' process, namely the distribution of the discrete 2D elements over the external surface of the CAD model of the component to be analyzed. Indeed, the position of the four vertices belonging to each rectangular panel is unknown a priori to the designer, especially in the case of complex freeform shapes. Consequently, the user has to sketch from scratch a cloud of points coherent with the surface of the object. However, this procedure is tedious, time-consuming and should be customized for each case study. FEM programs could be used to obtain a mesh of the component, but poor meshes can be obtained with complex bodies.

To overcome this limitation, this paper explores the use of the voxel-based representation [12] - defined as a volumetric geometry modelling using cubic cells - to define the position of the panel's vertices. Indeed, once the voxel model of the object of interest is obtained (V-model), the external voxelized surface can be easily extracted (EV-model). Then, the panelization can be sped up by matching each panel with the square surface of each voxel belonging to the EV-model. Thanks to a voxel-based panel method, the panel distribution process could be simplified and accelerated; however, the adoption of cubic cells on aerodynamic bodies creates a stair-effect on the external surface that may lead to fictitious peaks and valleys on the pressure distribution field that needs to be analyzed in depth. To the best of the author's knowledge, there is still a lack of scientific contributions dealing with voxels in fluid dynamic problems, especially about the possibility to explore their application to potential flow solutions. Indeed, just a few contributions have been retrieved from literature considering valid the voxel discretization for fluid dynamic purposes [15], [19].

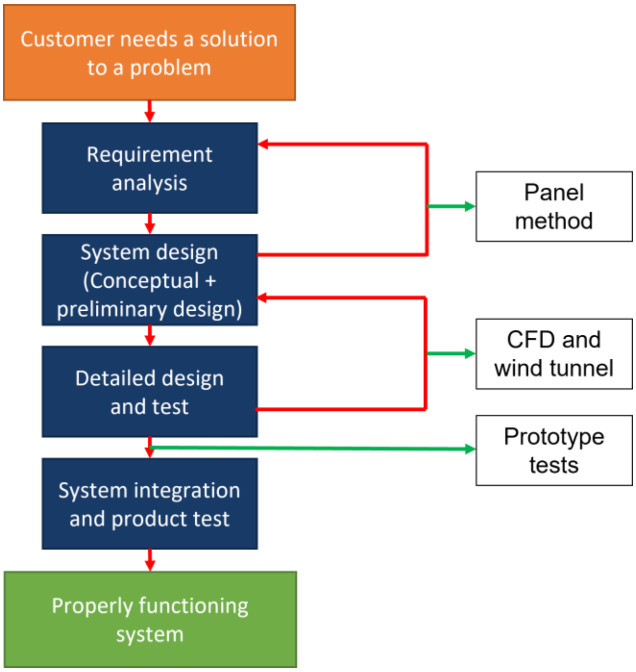

Figure 1: Typical design workflow; Aerodynamic approaches used on different design stages.

From a brief description of the numerical approaches available for the designers to estimate the aerodynamic loads, it is the opinion of the authors that a panel method is still nowadays a valuable approach; this is especially true for preliminary design stages. The push to reduce the environmental impact of transportation will lead in future times to consider the aerodynamics in conceptual design, also in cases where these issues have been overlooked in the past years. Modern concurrent engineering requires to consider just from the beginning of the development of a new product several aspects. In this framework, panel methods can play a role also nowadays. The scope of this paper is, therefore, to understand if the meshing procedures which is one of the main drawbacks of the panel method could be improved through a voxelization process: this is the original approach that is addressed and evaluated in this work. Indeed, the scope of this research is to preliminarily assess if such a methodology could provide the user with consistent results, reduce the workload and increase the industrial appeal of this simple but effective technique. To this aim, the voxel-based panel methodology has been applied to a benchmark body often used in the automotive aerodynamic field, the Ahmed Body [2]. However, the methodology could be applied also in other fields, such as trains, aircraft, and marine engineering. Several data from literature are available regarding the aerodynamic performances of the Ahmed body. Therefore, the voxel-based approach proposed in this research has been evaluated and tested, comparing the results obtained in terms of Lift and Drag with data from the literature. A voxelization is used to distribute automatically panels along the external surface of the body, while the open-source panel method software has been modified by authors to be integrated with the methodology. In this way, after setting the panel positions and features, the aerodynamic loads [10] have been computed. . What described in this paper can be seen as a further development of the activities envisaged in [3], where the same approach has been used to evaluate the lift coefficient of simple 2.5D bodies. The present paper aims to further evaluate the methodology and to develop new original strategies to extend the capabilities of the voxel-based panel method, preliminarily investigated in [3]. In more detail, in this paper, we further developed the methodology to make it possible the application of the voxel-based panel method to 3D models which can be obtained as a set of 2.5D bodies (2D shapes that are extruded in width). The main problem solved with this approach was represented by the definition of a mesh compatible with the panel disposition rules required by the panel method. It is worth noting that not all of the real-life 3D bodies can be subdivided into a set of 2.5D bodies. However, several objects can be studied, provided that a set of geometric adaptations depending on the shape of the body are implemented. The Ahmed body, which is a benchmark in automotive, represents an example of a 3D model that can be obtained as a set of 2.5D shapes. An approach to the estimation of the overall drag force is included as supplemental content compared to [3]: a comprehensive assessment of the errors obtained applying the methodology has been added too in this paper. Indeed, the final aim of the work herein described is to obtain a general-purpose fluid-dynamic tool that can be used in the conceptual design stage starting from a voxel representation of a body. This is to rapidly estimate the aerodynamics of 3D models, whose CAD model is available. Also, computer games could benefit from this methodology since voxel representation is widely used in this specific application, and sometimes accurate dynamics must be implemented. This methodology well fits the need for conceptual design and eventually heuristic optimization procedures because the 3D model of an object can be easily changed within a CAD package, using not only the user intervention but also using an automatic script or parametric models. In the further an automated procedure similar to that herein described can be followed to carry on aerodynamic analysis in short times. In this way, a lot of configurations can be compared to each other in short times, and also heuristic optimizations can be carried out considering thousands of different configurations.

The manuscript is organized as follows. Section 2 briefly describes the basic principles behind the panel method, and their coupling to obtain the voxel-based fluid-dynamics tool evaluated in this paper. Section 3 embeds a detailed description of the case study used to validate the proposed approach. A discussion of the results obtained is contained in Section 4. Finally, the work is summarized in Section 5 where conclusions and possible future developments of this research are listed.

2 MATERIALS AND METHODS

2.1 Open-source Panel Method

The authors selected APAME, an open-source panel method software, programmed in MATLAB that is available on the web [10] as a reference for the development of a panel code to be used in the research. It is worth noting that there are several open-source panel method tools available but, on the one hand, the majority is focused only on simple 2D geometries (i.e. [5] and [23]). On the other hand, panel methods developed for 3D models, such as Panair [6] developed by Boeing, does not allow easy integration of the voxel discretization. Other packages such as [7], are still in the development phase and it's quite difficult to understand the structure of the algorithms herein implemented. Compared to other software packages, APAME implements the panel method presented in [14]. In APAME, the panel method has been numerically implemented smartly, exploiting the routines of Matlab designed to handle the inversion of large size matrices to evaluate the aerodynamic loads and moments acting on entire complex structures. The implementation of the panel method suggested in APAME has been selected because of its capability to analyze 3D complex structures using easy discretization rules, and a high flexibility useful to interface the voxel-based discretization scripts. The panel method implemented in APAME can be useful in the case of the preliminary/conceptual design stage where calculation time is fundamental and only a rough estimation of aerodynamic lift and induced drag force/coefficient are sufficient. No corrections for viscosity are included. Therefore, due to the strong assumption of inviscid flow which is behind this approach, an accurate drag estimation is not possible: Therefore the implementation of the panel method presented in APAME has been improved to compute not only the induced drag but also the skin drag which depends on viscous effects and is not captured by the panel methodology. Moreover, also the shape drag (whose effect is considered in the case of blunt bodies) isn't considered in the panel methods. Due to flow assumptions, all the potential flow approaches shouldn't be considered reliable to estimate drag, unless geometrical corrections based on equivalent flat plate skin and drag shape are included. Taking in mind these limitations, the implementation of the panel method illustrated in APAME has been evaluated as a very good baseline to allow the development of a code useful to substitute CFD programs for subsonic attached regimes to estimate lift and induced drag of simple bodies in the preliminary design context.

As previously mentioned, the computational demand to solve potential flow problems is lower by far than CFD approaches because the discretization occurs only on the external surface of the body instead of subdividing a volumetric flow domain larger several times compared to the dimensions of the body to analyse. The elementary features that discretize the object are called panels, and they have the shape of a segment in 2D problems, and usually a rectangle in the case of 3D bodies. To solve the flow, the panel method relies on the distribution of elementary singularities (potential source, dipole, sink, vortex) on each panel to satisfy the boundary conditions determined by the problem, such as the no-slip condition at the walls or the so-called Kutta condition at the trailing edge of lifting bodies such as airfoils. The solver takes into account the interaction between the distributed singularities, and far from the object, it assures the free stream flow condition. The solution can be found using the superposition principle, meaning that the sum of singularities of the governing equations forms a solution as well. In the panel method suggested in [14] and implemented in the APAME specific case, quadrilateral panels with a constant distribution of point sources and doublets (equivalent to vortex ring) are considered. The panel induced potentials on a spatial point x, y and z are computed in eq. 1, where σ is the source strength, μ the dipole strength and S is the panel surface.

$$\begin{cases} \Phi_{source}(x,y,z) = \dfrac{-\sigma S}{4\pi}[x^2+y^2+z^2]^{-\frac{1}{2}} \\ \Phi_{dipole}(x,y,z) = \dfrac{-\mu S}{4\pi} z[x^2+y^2+z^2]^{-\frac{3}{2}} \end{cases} \tag{1}$$

Before finding the velocity and pressure distributions, the panel method evaluates the source and dipole strengths distributed along the 3D body along with the influence coefficient matrices $\boldsymbol{a}$ and $\boldsymbol{b}$ of such singularities. The equation useful to solve the potential flow is based on the Dirichlet's condition [18], computed considering all the O number of panels, saying that there is no flow inside the body (eq. 2).

$$\sum_{i=1}^{o} a_i \mu_i + \sum_{i=1}^{o} b_i \sigma_i = 0 \tag{2}$$

To solve the flow, the classical panel method implementation suggests using the simplified version of the Navier Stokes equations. In particular, the continuity equation becomes the Laplace equation (eq. 3), after the simplification due to the flow assumptions.

$$\frac{\partial^2 \varphi}{\partial x_i^2} = 0; \ \frac{\partial \varphi}{\partial x_i} = v_i \tag{3}$$

The solution of the problem can be identified as φ that is the scalar speed potential of the velocity field v_i. Moreover, to completely solve a flow around an object, it is necessary to estimate even the pressure distribution. Thanks to the simplified approach offered by the panel method, the pressure distribution p can be estimated using the Euler-Bernoulli equation (eq. 4), once the velocity field is known, being ρ the density of the medium.

$$\frac{v^2}{2} + \frac{p}{\rho} + gz = cost \tag{4}$$

At the end of the calculations, with the panel method, it is then possible to compute the aerodynamic forces and moments by integrating the pressure distribution over the external surface of the tested body. Further details about the panel method theory selected for this work [14] and the numerical implementation of it in APAME can be found in the references [10] and [11].

2.2 Voxel-based Modelling

In this paragraph, the voxel representation technique initially developed in [3] is described in more detail, along with a graphic flowchart showing the steps of the volumetric discretization approach implemented for the sake of this research.

The voxel-based modelling is a CAD 3D representation technique based on the use of unit cubic cells called voxels which can be seen as the equivalent of the pixels in 2D images. Voxelization is mainly used to speed up geometrical and algebraic manipulation of complex shapes. Compared to the Boundary representation (B-rep) used in commercial Computer-Aided Design (CAD) software, the voxel modelling translates a 3D model into a logical 3D matrix (black and white representation) where each value represents every single voxel [24]. Each elementary cell is given a binary value depending on the spatial position of the voxel: if an element belongs to the 3D model, the value 1 is assigned, 0 otherwise. According to the available literature, Voxel-based modelling is mainly used to accelerate geometrical manipulations because graphical operations become just a matter of matrix operations. Moreover, with voxels, it's possible storing details of intricated 3D model interiors, a feature that is not possible with B-rep. Indeed, the common CAD software packages use the B-rep and only store information regarding the external surface shape. As a consequence, voxelization is mainly used in engineering, medicine, geography and computer science to provide for an alternative modelling technique in the case of small and complex models. Moreover, voxel discretization rarely fails compared to other geometrical discretization based on more complex unitary elements. Depending on the voxel resolution, the annoying stair-effect of the external surface of V-models can be mitigated. Figure 2 shows the effect of voxel size on the representation of a real component to convince the reader about this feature.

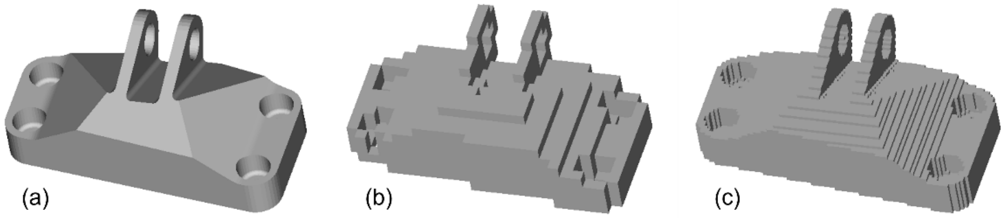

Figure 2: GE bracket [25] model using: (a) standard B-rep visualization, GE bracket V-model with coarse resolution (b) and GE bracket V-model with fine resolution (c).

Among the wide panorama of voxelization algorithms, such as volume sampling, ray-stabbing or parity count [8], the ray-tracing intersection method [20] is used in this research to complete the innovative voxel-based fluid-dynamics approach. The ray-tracing intersection algorithm was programmed in MATLAB language, and it has been modified in this study to manage voxel-based models. To start the voxelization process, the user needs the Standard Triangulation Language (STL) 3D model of the component to test and to choose the voxel resolution d that directly affects the voxel spatial grid S_{X_i}. Then, the algorithm ray-traces in all the three main directions (x, y and z) and combines the results coming from the different dimensions. Specifically, for each triangle of the STL file, the voxelization follows the succeeding steps: a) to determine the edges of the triangle; b) to compute the opposite vertex of the selected edge; c) to find the ray relative to the selected edge; d) to check if the relative ray is on the identical side of the selected edge; e) if the check is positive for all the edges of the selected triangle, then it is undoubtedly true that the ray flows through the facet, and for this reason, the matrix element should be activated. For sake of clarity, a flowchart of the voxelization process followed in this paper is shown in Figure 3 to provide the reader with a list of the single phases. Thanks to a reliable algorithm implemented in MATLAB, the voxelization algorithm previously described is combined with the panel method software to speed up the 'panelization' process. Indeed, the methodology herein described is based on the use of the square external surface of the Voxel model as the elementary panels to be used in the potential-flow solver to get the aerodynamic loads. Compared to the classic panel method approach, the voxel exploitation reflects on a user-friendly technique, and definitely on an automated tool for the preliminary solution of aerodynamic performances of selected bodies. This automated process is so far limited for all the 3D models which can be decomposed as a sequence of 2.5D geometries to be compatible with the numbering and distribution of the panels over the surface. This study can be seen as an original continuation of what was described in [3] where only simple 2.5D bodies had been treated: in this paper, a more general-purpose 3D tool has been developed, useful to treat 3D models more significant for the industrial engineering. In particular, 2.5D geometries can be defined as a 2D silhouette that is extruded in the third dimension. Further studies will enlarge the applicability of this approach to all the 3D shapes.

2.3 Voxel-based Fluid-dynamics Methodology

In this section, the overall methodology combining the voxel-based modelling and the panel method software will be described in detail, with the support of Figure 4, where the flowchart of the developed approach is depicted. More in detail, the developed approach starts with the availability of the 3D model of the object of interest (i.e. a wing or a car) saved in the STL format (Step 1 of Figure 4) that can be obtained with the largest part of commercial CAD software. Then, the algorithm extracts the matrices describing the triangulated mesh discretization contained in the STL file. In particular, the matrices of vertices V, of facets F and the matrix of normal vector components are computed (Step 2 of Figure 4). Being available this information, the user is asked to actively intervene (orange arrow in Figure 4) to choose the voxel resolution (vox_{dim}) along with the X and Z directions, and the resolution in the spanwise Y direction, according to the frame of reference (f.o.r.) shown in Figure 5.

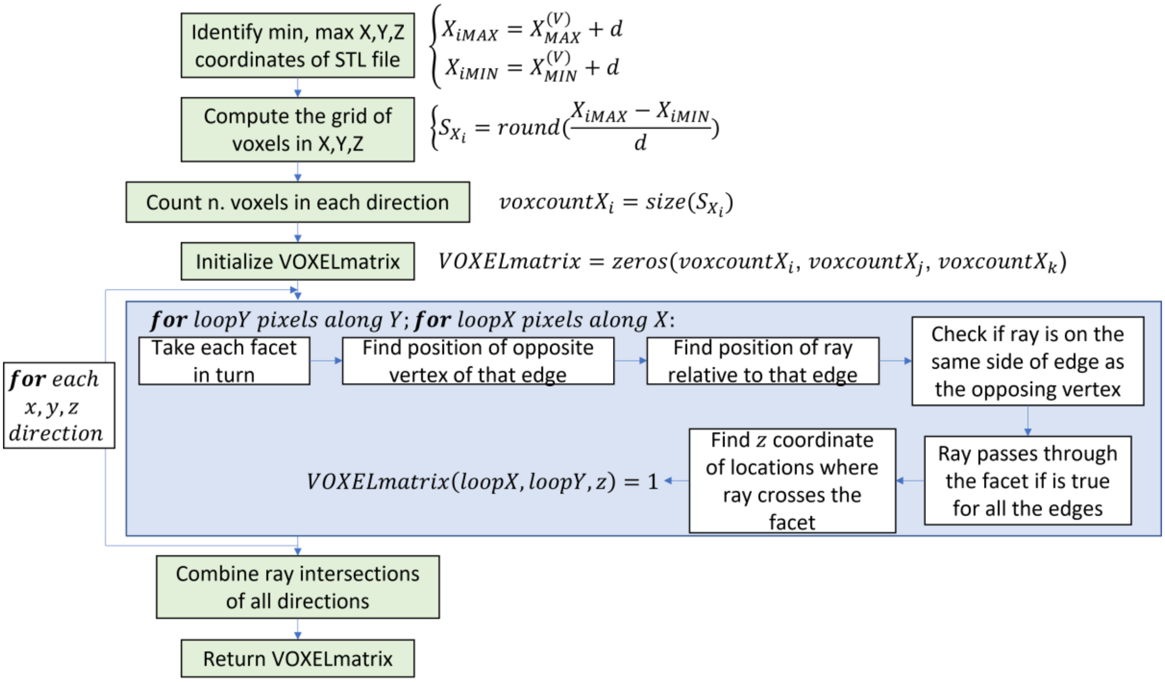

Figure 3: Flowchart of ray-intersection voxelization methodology.

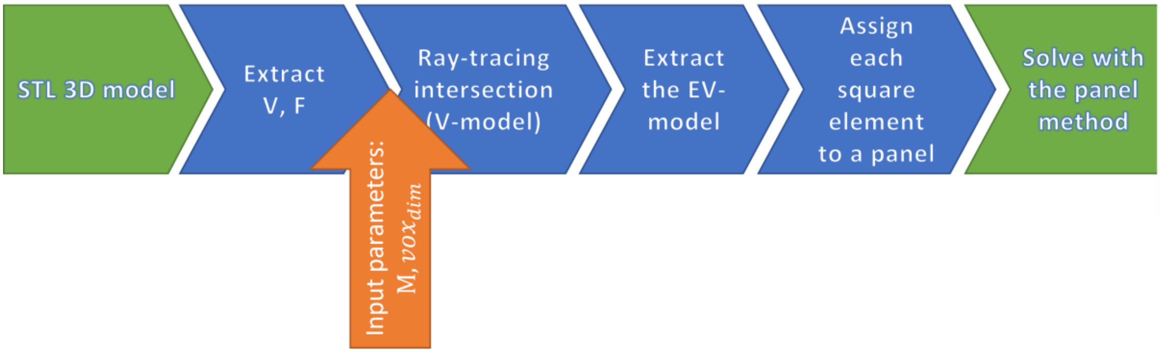

Figure 4: Graphical representation of the methodology behind the voxel-based fluid-dynamic approach.

The setting of the voxel resolution must be implemented because in the preliminary design stages the complexity of the 3D bodies should be considered when the size of the voxels is set. Moreover, a further simplification can be carried out in the case of linearly extruded 3D bodies (e.g. straight wing) or models that can be decomposed into many 2.5D models: these kinds of shapes can be seen as a 2D silhouette in the XZ plane that is extruded in the third direction (in this case the Y direction).

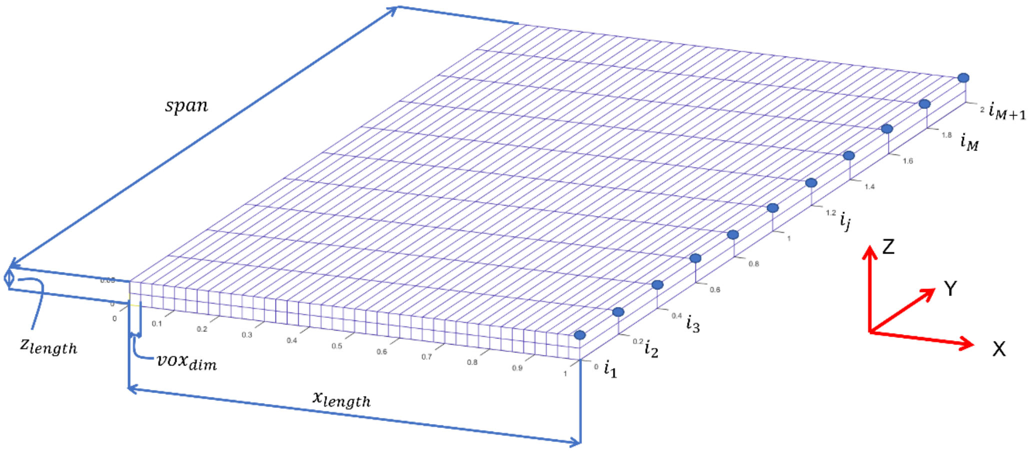

Figure 5: V-model of a flat plate; definition of the frame of reference to be compatible with APAME software.

Additionally, according to the panelling convention suggested in APAME, the voxel resolution along X and Z affects the number of lateral panels N, while the resolution in the Y direction affects the number of longitudinal panels M. Indeed, the voxelization resolution along the 3 directions can be computed as (eq. 5):

$$\Delta y = \frac{span}{M}; \qquad vox_{dim} = \Delta x = \Delta z \tag{5}$$

The choice of the f.o.r is accomplished to require simple changes in the panel method implementation selected: this to make the overall approach as efficient as possible. Once the voxel resolution is set, these input parameters are sent to the voxelization function that implements the ray-tracing intersection algorithm, described in the previous section (Step 3 of Figure 4). The function can discretize the 3D model and convert it into a V-model, returning a logical 3D matrix of dimensions n x m x M, where:

$$n = \frac{x_{length}}{\Delta x}; m = \frac{z_{length}}{\Delta z}; M = \frac{span}{\Delta y} \tag{6}$$

Being available the V-model, the proposed approach continues by extracting the EV-model, which is composed just by the external surface of the voxel model (Step 4 of Figure 4). The extraction of the EV-model is performed based on a MATLAB function available to the developers' community [1], and the function returns the matrix of vertices coordinates of square surface elements. Before sending the EV-model to the panel method software it is required to assign each square element to a panel, which can be considered as the elementary unit necessary to solve the potential-flow problem (Step 5 of Figure 4). This process is carried out rearranging the matrix coming from Step 4: the spatial coordinates of the EV-model are reorganized in a specific order, according to the convention proposed in APAME and saved into a MATLAB structure variable, called mesh:

- mesh.x: an (N+1)x(M+1) matrix taking in the N+1 x coordinates of the vertices of the EV-model, collected in Clock Wise order (CW) in each column. The M+1 columns describe the M+1 sections along the Y direction;
- mesh.z: an (N+1)x(M+1) matrix taking in the N+1 z coordinates of the vertices of the EV-model, collected in Clock Wise order (CW) in each column. The M+1 columns describe the M+1 sections along the Y direction;
- mesh.y: an (N+1)x(M+1) a matrix comprising a spatial grid in the lateral (or spanwise) dimension with M + 1 y coordinates in each row, which are repeated N + 1 times to depict the lateral (or spanwise) dimension extrusion.

Once the external surface panel structure is arranged, the variable becomes the input of the panel method (Step 6 of Figure 4). This algorithm can return the pressure coefficient and velocity distributions together with the aerodynamic forces and moments acting in all the 3 directions. Before running the simulation, it is fundamental to properly set the free stream velocity module v_∞, the density of the medium and the direction of v_∞ in terms of angle of attack (α) and sideslip angle β and the reference surface S_{ref} that should be used to compute the aerodynamic load coefficients, according to eq. 7, being q the reference dynamic pressure. This research will focus only on the lift and drag estimation of a V-model, therefore only a longitudinal analysis is performed; also lateral-directional effects could be computed, but this possible implementation goes beyond the scope of this research. For that reason, the lift and drag coefficients are computed according to eq. 8.

$$C_X = \frac{F_X}{qS_{ref}};\, C_Z = \frac{F_Z}{qS_{ref}} \tag{7}$$

$$C_L = C_Z\cos(\alpha) - C_X\sin(\alpha);\; C_D = C_Z\sin(\alpha) + C_X\cos(\alpha) \tag{8}$$

Thanks to the coupling of the panel method with the voxelization algorithm, the user obtain automatically the distribution of panels conformal to the external surface of the body he/she would like to analyze. Thus, the tedious step of designing from scratch in a CAD quadrilateral mesh coherently with the surface of the geometry is completely automated, making this approach useful in the preliminary design stages, as the computational time dramatically decreases. As already mentioned, the computational reduction of the configuration analyses is of straightforward importance in the first design stages where several configurations, such as different kinds of airfoils or shapes, should be rapidly tested to find the optimum one.

In the following section, the authors present a case study of relevance for the scientific community to validate the proposed approach: the methodology is applied to the Ahmed Body 3D model. Furthermore, the results from the analysis are compared with values coming from literature references. In other words, the voxel-based fluid dynamics approach is used to determine the lift and drag forces acting on the body. As already pinpointed, the drag is evaluated using geometrical corrections based on equivalent flat plate skin and drag shape because the panel method is able only to estimate the induced drag source: in blunt bodies, shape drag could play a sensitive role in the overall drag, while in the case of streamlined bodies skin effects can be important in case of a small lift where induced drag is negligible.

3 RESULTS

A case study has been tested to validate the proposed approach and to study how the voxel discretization may affect numerically the aerodynamic performances in the analysis of a 3D body. Moreover, a more in-depth description of the methodology to evaluate the overall drag force is included in this contribution compared to [3] and a 3D case study relevant for automotive engineering has been analyzed. Concerning this point, the authors selected the Ahmed Body as the main case study to investigate: because several aerodynamic studies are available in the scientific literature. This is true to the extent that the Ahmed body is often taken as a benchmark for aerodynamic studies. This 3D model was initially designed by S. R. Ahmed in 1984 [2] to study the effects of the slant angle on the wake of a simplified model that tries to imitate the behaviour of more complex cars. The dimensions of the original Ahmed body 3D model are included in Figure 6 which has been sketched by authors to provide the reader with the dimensions used in this study.

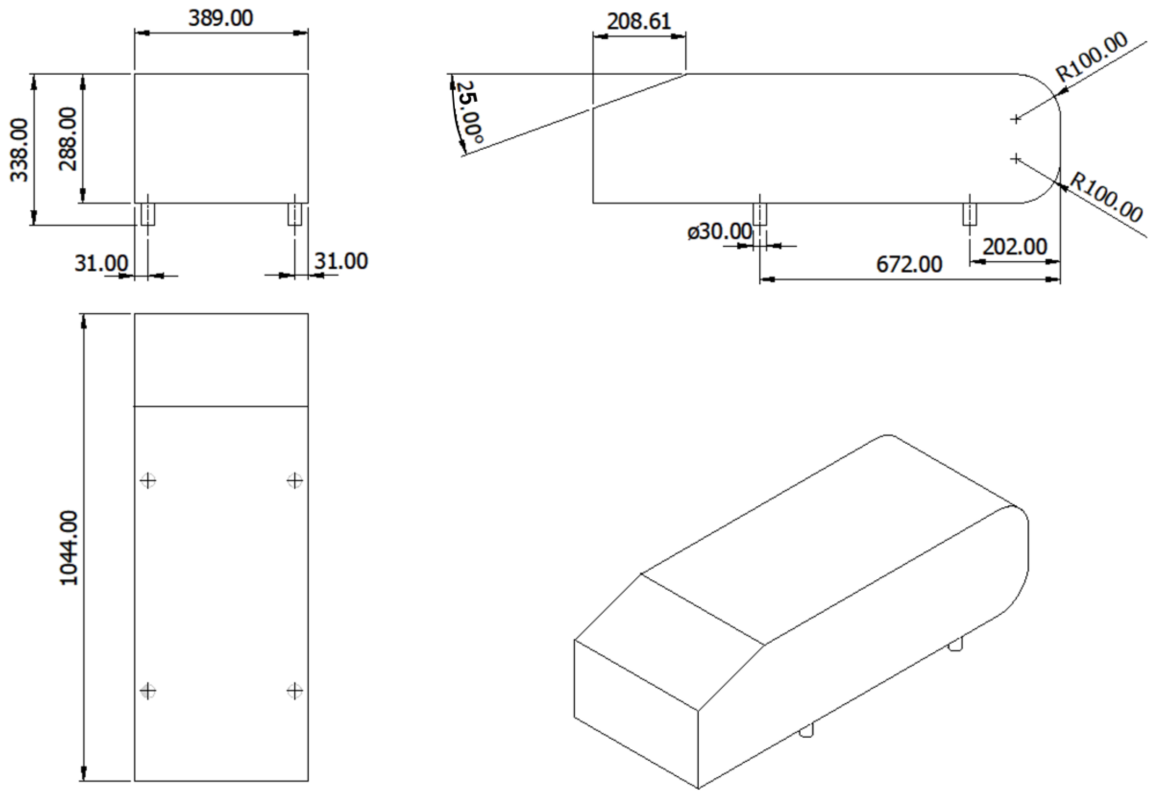

Figure 6: Ahmed body 3D model used as case study to validate the proposed voxel-based fluid dynamics approach.

This research aims to develop a fluid dynamics tool able to accurately estimate both lift and drag forces of a 3D model, so the authors chose the study performed by Dobrev and Massouh [9] as a reference. On the one hand, in that source, there are numerical references both for lift and drag aerodynamic coefficients which have been evaluated through numerical and experimental tests. On the other hand, the majority of the other references consider only the drag estimation. In any case, the drag value estimated by Dobrev and Massoush is in line with some of the more cited references by the whole scientific community, such as Krajnovic and Davidson's study [16], and Katz's book [13].

The performances of the voxel-based panel method are evaluated in terms of estimation of aerodynamic efficiency E defined as the ratio between the lift and the overall drag loads (eq. 9).

$$E = \frac{L}{D_{tot}} \tag{9}$$

It is important to pinpoint that D_{tot} stands for the total amount of the aerodynamic drag. In aerodynamics, the total drag is defined as the sum of 3 contributions:

- Skin friction drag, caused by the viscosity of the medium;
- Form drag, caused by the shape of the body;
- Induced drag, caused by the lift of the object that redirects the airflow coming at it. It affects all the bodies that generate lift/downforce.

The study of Dobrev and Massouh [6] mainly focuses on the investigation of the lift and drag relationship when the slant angle of the Ahmed body changes from 0° to 90°. The survey is carried out both experimentally, using a wind tunnel facility, and numerically with CFD using Ansys Fluent software by varying the slant angle. For this work, only the results coming from the simulations involving the Ahmed Body with a 25° slant angle are taken as a reference to validate the approach described in the present contribution. The boundary conditions used in Dobrev and Massouh's study are:

- Free stream velocity: 30 m/s;
- Angle of attack: 0°;
- Air density: 1,225 kg/m^3;
- The reference area is set to be the cross-section of the Ahmed body.

The aerodynamic lift and drag benchmarking values of the Ahmed Body with a 25° slant angle coming from [9] are collected in Table 1 for reference. To fit the test chamber of the wind tunnel facility, the authors had to scale down the 3D model with a ratio of 3:4. In the following, to evaluate the accuracy of the proposed approach, only the experimental aerodynamic coefficient values will be taken as a reference value. The performance comparison will be computed as a percentage error between the values coming from Dobrev's study and those coming from the methodology herein developed.

	Experimental	**CFD**
Drag [N]	11.13	10.60
Lift [N]	7.79	8.38
Efficiency	0.7	0.79

Table 1: Lift and Drag reference values of the Ahmed Body with a 25° slant angle. Data from reference [9].

3.1 Voxel Resolution Independence Study

To understand how the voxel resolution may affect the results of the proposed algorithm, a mesh independence study has been set up. Both the input parameters vox_{dim} and M are analyzed. To achieve the best input configuration, several simulations have been set up using the voxel-based fluid dynamics approach, fixing one parameter and varying the second one. This study is feasible since the computational time and power needed to run a single simulation ranges from a few seconds to minutes thanks to the simplicity of the potential flow methodology. The whole set of simulations has been run on a personal laptop with an i7 4-core 2.6 GHz CPU and 16 GB of RAM. Figure 7 shows the effect of voxel size on the representation of the Ahmed body.

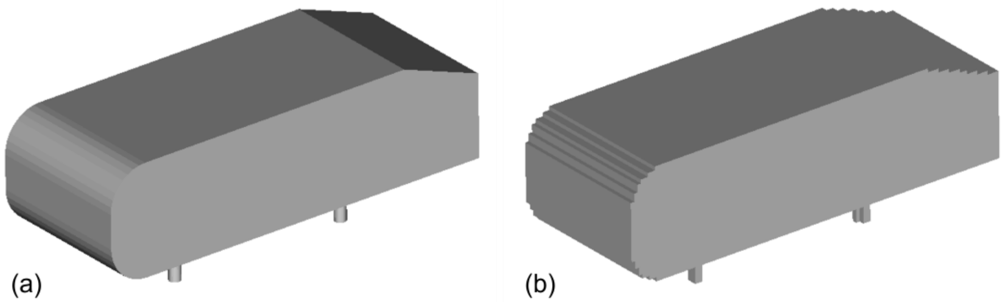

Figure 7: (a) Ahmed body 3D model using B-rep in CAD and (b) the same 3D model after the voxelization.

Starting with the analysis of the voxel resolution vox_{dim} in the XZ direction, various simulations have been set up using different values for vox_{dim}, while M is fixed to 30. Figure 8 shows the behaviour of the estimation error for the lift coefficient C_L with a blue line and how this parameter affects the overall computational time, in red.

It is straightforward noting that using a finer voxel discretization the computational time required to find a solution exponentially increases, while the estimation error of the results is asymptotically zeroed. Thanks to this set of simulations, it is possible to find the coarser voxel resolution that still gives satisfactory results (less than 2% of error) without compromising the computational speed of the voxel-based approach (green dot line in Figure 8). The identified value of vox_{dim} is around 2% of the overall length of the 3D model in the X-direction. In the specific case of the Ahmed body that is 1.044m in length, the best XZ voxel resolution is 0.02m. A total of 27030 voxels are used to discretize Ahmed's body in this case.

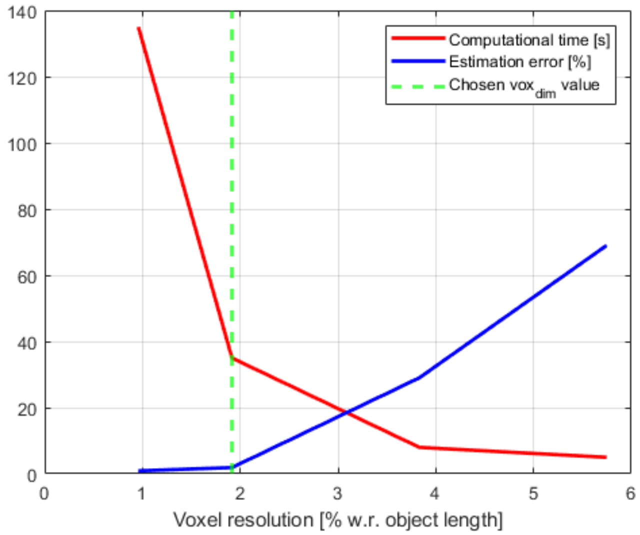

Figure 8: Computational time (in red) and estimation error for the lift coefficient (in blue) of the Ahmed body, varying the voxel resolution in the XZ direction, is given as a percentage of the overall length along the X-direction.

A similar approach is used to find the best longitudinal voxel resolution M. Once more, a set of simulations has been carried out changing the second input parameter, while keeping fixed vox_{dim} to the value found from the previous study. The computation time and the accuracy of the results are collected in Figure 9. The results demonstrate once again that a finer resolution affects mainly the computational time, without affecting the accuracy of the results. For this reason, a mid-resolution ($M = 30$) can be a good choice for further studies. From Figure 9, it seems that with a lower amount of lateral (or spanwise) panels a satisfactory result can be achieved with less computational effort. However, the best discretization resolution is a trade-off between accuracy, computational burden, and geometry consistency. Indeed, the designer should also verify that the voxel resolution is sufficiently accurate to capture the model's characteristics. Based on several simulations carried out by the authors, $M = 30$ has been set, being the lowest value that allows a consistent geometry with the original one, especially in the lowest part of the body where the four cylinders stand for the car's wheels.

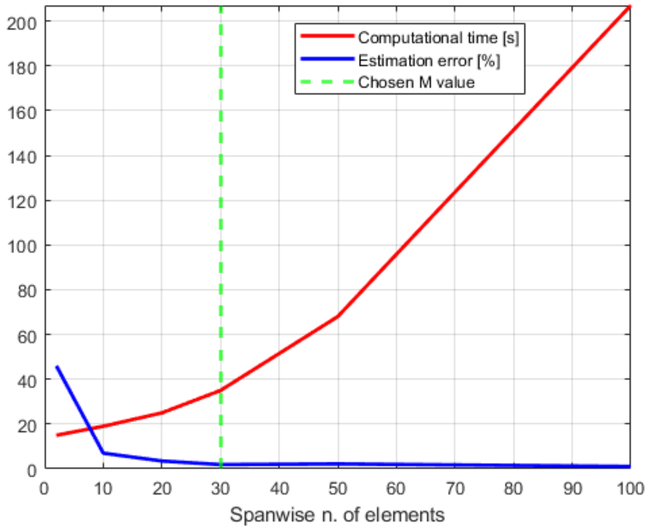

Figure 9: Computational time (in red) and estimation error for the lift coefficient (in blue) of the Ahmed body, varying the voxel resolution in the Y direction, given as the overall number of panels M along the Y direction.

3.2 Validation Test

Once the input parameters are fixed, a single simulation is set up using the same boundary conditions chosen in [9]. Starting from the overall interval of time necessary to set up the numerical experiment of the proposed approach, MATLAB took 14 seconds to automatically distribute the panels along the external surface of the 3D model that can be seen as the meshing stage. This result is encouraging compared to classic panel method approaches available in the literature where the designer has to design from scratch the panel distribution coherent with the external surface of the 3D model whose CAD model is available. Next, the panel method took 21 seconds to evaluate the influence coefficients of the different singularities distributed along the body and 2 seconds to solve the problem. Summing up all the timings, a preliminary solution can be found in less than a minute. This time is lower by far if compared to experiments carried out in wind tunnel facilities which may require even weeks to be set up and to analyse data. Moreover, also CFD numerical approach takes a longer time (usually several hours) to solve the aerodynamic problem because volumetric meshing should be done to discretize the overall flow domain and to locally solve the Navier Stokes equations. Moreover, with CFD the meshing phase usually requires a heavy and painstakingly human intervention with several hours spent in the detection of inconsistent zones of the mesh. For this reason, the proposed voxel-based panel method could be of valuable importance provided that satisfactory results in terms of accuracy can be achieved: the following of this chapter will therefore deal with the precision of the results.

To understand the methodology accuracy, a final simulation is carried out using $M = 30$ and $vox_{dim} = 0{,}02m$:5894 panels are obtained to discretize the Ahmed body (Figure 10a), which is by far computationally lower than the 106 tetrahedral elements used in [17]. After the run, the pressure and velocity field distributions are extracted and graphically shared with the designer (respectively Figure 10b and 10c).

By integrating the pressure distribution over the surface of the model, the panel method code can compute the overall lift and induced drag loads. Moreover, thanks to geometrical shape corrections, it is possible to roughly estimate the skin friction drag coefficient C_f using the Prandtl one-seventh power-law for turbulent flows [21] which is calibrated experimentally [22] (eq 10.). The estimated C_f depends on the Reynolds number that for the specific case study can be evaluated knowing the free stream velocity module V, the characteristic dimension of the 3D model x_{length}, the medium density ρ and the kinematic viscosity μ (eq. 11).

$$C_f = 0{,}0592 \cdot Re_x^{-\frac{1}{5}}; \quad 5 \cdot 10^5 < Re_x < 10^7 \tag{10}$$

$$Re_x = \frac{V \cdot x_{length} \cdot \rho}{\mu} = \frac{30 \left(\frac{m}{s}\right) \cdot 1{,}044\ (m) \cdot 1{,}225\ \left(\frac{kg}{m^3}\right)}{15{,}52 \cdot 10^{-6}\ \left(\frac{kg}{m \cdot s}\right)} = 2{,}56 \cdot 10^6 \tag{11}$$

In the case of low Reynolds number, semi-empirical formulas to estimate the drag with laminar flows can be used. The form drag coefficient can be roughly estimated by looking at the blunt bodies drag coefficient tables widely available in the literature, assuming the Ahmed body close to a long cylinder [4]. Thanks to the previous assumptions, the total drag and lift loads along with the aerodynamic efficiency can be computed by multiplying the aerodynamic coefficients by the dynamic pressure and the reference area. All the results obtained in this study are collected in Table 2.

	Aerodynamic Load [N]	**Computed by**
Lift	55.882	panel method
Induced drag	10.517	panel method
Skin friction drag	2.346	[21]
Form drag	63.057	[4]
Total drag	75.92	panel method + corrections

Table 2: Lift and Drag computed values for the Ahmed Body with the voxel-based panel method.

4 DISCUSSION

In this section, the results collected in the previous unit are discussed based on the perspective of reference studies, such as [9]. As previously stated, the first impressive result is the computational time necessary to achieve a numerical solution with the new voxel-based panel method. Just a few seconds are sufficient to get a complete overview of velocity and pressure distributions and to evaluate the aerodynamic forces and moments along the three axes. Such a fast computation perfectly fits the conceptual design stages, as can be seen in Figure 1 where a possible iterative design procedure is presented. After a change in the 3D model, new lift and drag results would be obtained in a short time, thus making possible trade-off studies or sensitivity analyses. Also in the case of heuristic optimizations based on evolutionary or swarm algorithms the whole time to reach convergence would be hours.

The voxel resolution affects the rapidity of the computation: a finer mesh reflects on longer simulation time, but more accurate results. However, Section 3 includes some guidelines on how to choose proper input parameters for the best panel discretization. Moving towards the main results, it was widely explained that using the voxel-based panel method it is possible to evaluate only the lift and induced drag sources.

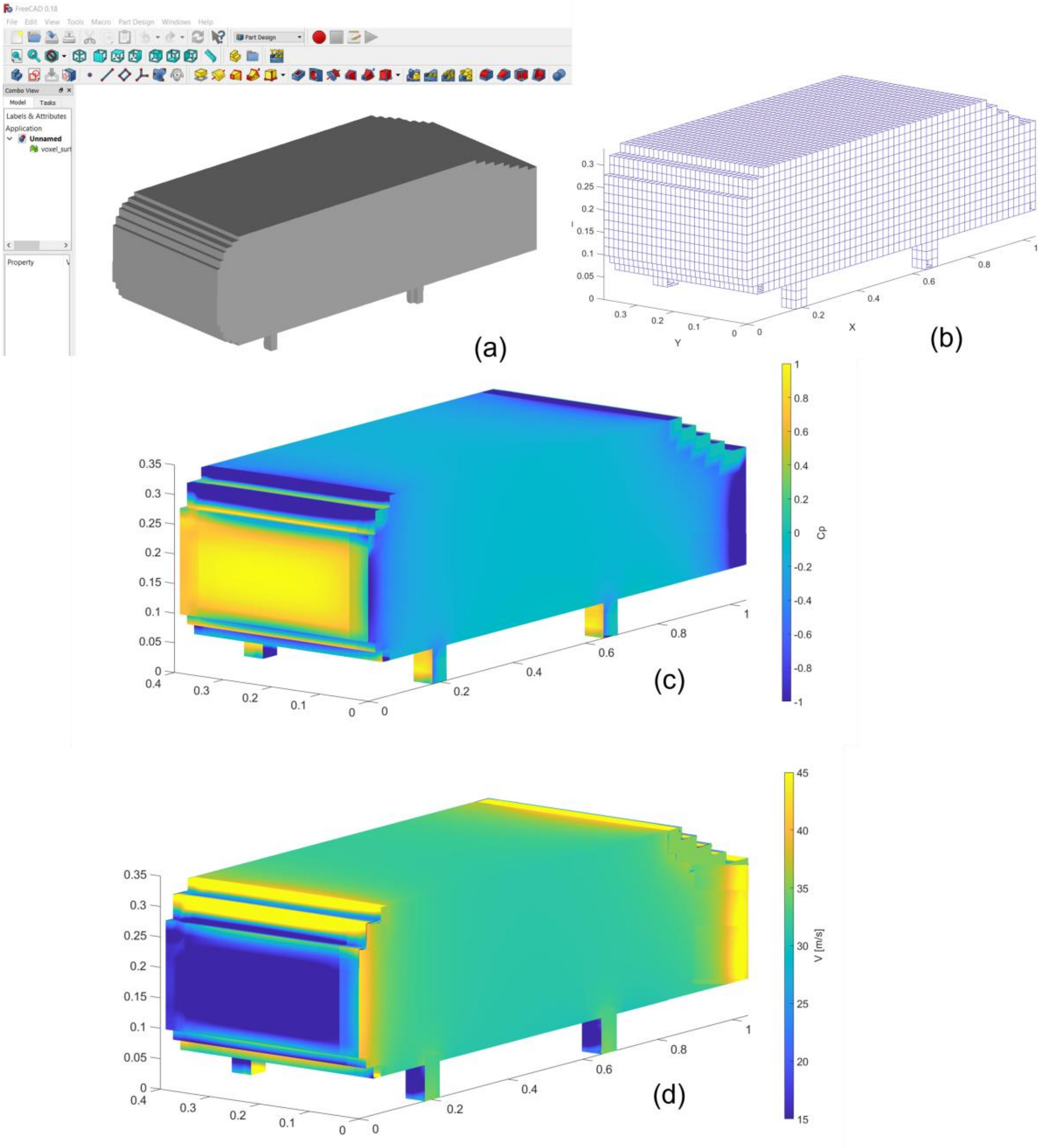

Figure 10: (a) CAD model of the voxelized Ahmed body in the FreeCAD environment; (b) panels distribution on the Ahmed body; (c) pressure coefficient distribution over the Ahmed body; (d) velocity field distribution over the Ahmed body.

However, by applying simple geometrical shape corrections and taking some reference values from the literature, it was possible to compute in a very simple and straightforward way the remaining shape and skin drag sources and so estimating the overall drag (in the following, called approach A). In terms of aerodynamic efficiency, the result obtained by the panel method plus the geometrical corrections is encouragingly close to the reference value coming from the literature.

Indeed, an alternative approach to estimate the overall drag avoiding the shape corrections could be to evaluate the lift from the voxel-based panel method software; then, using the aerodynamic efficiency value coming from the literature [9] ($E = 0.79$) assumed as a benchmark, the overall drag could be computed by dividing the lift times the aerodynamic efficiency (called approach B in the following). However, this second approach could be followed only for bodies where literature suggests values of efficiency in working conditions similar to that under investigation. Moreover, with approach B, the designer does not have information about the skin and shape drag estimation, which could be limiting in some circumstances. For streamlined shapes, shape drag can be neglected, while for blunt bodies friction drag is usually negligible. The comparison of both approaches is included in Table 3, along with numerical values. In Table 3 the blue arrows describe the workflow of both compared approaches, to convince the reader that the one evaluated by authors is more straightforward and consistent.

	A	B
Lift [N]	55.882	55.882
Total drag [N]	75.92	70.737
E	0.736	0.79
err_{Drag} [%]	7.35	/

Table 3: Comparison of total Drag estimation approaches: approach A involves the geometrical corrections, while approach B computes the drag knowing the efficiency reference value from literature. The blue arrows graphically show the design workflow followed by each approach.

It can be easily seen that the more straightforward new approach introduced and assessed in this paper is characterized by a small and acceptable evaluation error ($\cong 7\%$) when the overall drag should be computed compared to the more rigorous B approach. Both A and B approaches, coupled with a fast and accurate voxel-based panel method, have not been investigated either in [3] or in the relevant literature. Indeed, it is of fundamental importance to compute both lift and drag aerodynamic force with as low as possible computational effort, also in the conceptual design stage.

To conclude, in this paper a fast and easy to be implemented approach has been described, based on the combination of the panel method with the voxel discretization. Indeed, voxelization mitigates the negative aspects of a tedious and manual panelization process or the use of external meshing software packages. Moreover, a methodology is described to compute the overall drag force: only the generated lift load is necessary because geometrical corrections cope with the lack of panel method in the computation of skin and shape drag. The results show encouraging agreement with reference values coming from literature. Moreover, an original application of voxel has been proposed and investigated in this paper, and it could open to new studies in this field.

5 CONCLUSIONS

The potential flow approaches and in particular the panel method are still valuable methodologies to numerically solve aerodynamic problems in the context of the preliminary design stages for subsonic flows. Compared to CFD, wind tunnel, and experimental tests on full-scale models, the approach described by the authors can decrease the computational effort by far: satisfactory results in terms of lift and drag estimation can be achieved. However, the main challenge of the panel method is the positioning of panels. They can be defined as the discrete elements used in the panel method numerical approach in which a potential flow singularity must be associated. This process has to be done manually by the user who must mesh the body in a way coherent with the external surface of the 3D model, exponentially increasing the design effort, especially for complex freeform models. As an alternative, external meshing software could be used, thus requiring a further step and possible manual intervention also in this case to set the mesh.

To mitigate this aspect, the authors proposed and evaluated in this paper a numerical methodology combining the panel method with voxelization to automatically distribute panels on the surface of the model. Indeed, each panel collapses with the external square surface of each voxel belonging to the EV-model. Then, the spatial coordinates of the square panels are rearranged to be compatible with a panel-based software developed by authors, based on the implementation suggested in an open-source panel method [10].

To validate the proposed methodology, a case study based on the benchmarking model of the Ahmed body is set up. Reference values in terms of both lift and drag are selected among a wide panorama of scientific contributions in the literature. A parametric study has been included to demonstrate how the voxel discretization affects the results. Moreover, some guidelines to optimize the voxel dimensions to get the best panelization have been included as well. The panel method is known to have limitations about the overall drag estimation because of the strong flow assumption on which the approach is based. To overcome this second challenge, an approach based on simple calculations is proposed to compute in a simplified way the overall drag using corrections for skin drag and form drag coming from tables from literature. This approach demonstrates to be valid in terms of accuracy of the results compared to literature values (less than 10% of error in overall drag estimation), and computationally faster than classical CFD analyses.

As a final comment, it is worth noting that using this approach, the analysis of a body is highly automated, thus requiring only a CAD model. All of the commercial CAD software are equipped with the capability to save STL models both starting from a single part and forming an assembly. Therefore, fluid dynamics analyses could be carried out directly from aesthetical/conceptual 3D models developed by design departments, without the need for operators skilled in meshing and CFD analyses. Another interesting point addressed in this paper is how a body represented in voxel representation can be analysed to obtain fluid dynamic information without the need for translation into other CAD representations and further meshing. The main limit of the methodology is that the panelization procedure is automated for 3D bodies that may be approximated with a set of 2.5D bodies (2D shapes that are extruded in width), while for 3D bodies tailored algorithms must be developed to translate complex panels distribution into a format which could be read by the currently implemented panel method algorithms: complex bodies must be divided into two kinds of sub-parts. For the lifting sub-parts (typically showing sharp trailing edges), the Kutta condition must be adopted, while for non-lifting sub-parts this is not required. On the other hand, several real-life bodies can be decomposed into a set of 2.5D bodies, so that also in this present form the methodology can be applied to objects with a complex shape with a high significance for industrial engineering.

Soon, the voxel-based panel method will be enhanced by further studies to simulate complex and fully 3D shapes to understand how the automatic voxel panelization behaves in more challenging contexts. Moreover, the results should be compared with other numerical methods (i.e. CFD) when literature reference values are not available.

Antonio Bacciaglia, https://orcid.org/0000-0002-4384-6300
Alessandro Ceruti, https://orcid.org/0000-0001-7947-3932
Alfredo Liverani, https://orcid.org/0000-0002-3255-9381

REFERENCES

[1] Adam, A.: Converting a 3D logical array into an STL surface mesh, MATLAB Central File Exchange, 2021.

[2] Ahmed, S. R.; Ramm, G.; Faltin, G.: Some Salient Features Of The Time-Averaged Ground Vehicle Wake, presented at the SAE International Congress and Exposition, 1984, 840300. https://doi.org/10.4271/840300

[3] Bacciaglia, A.; Ceruti, A.; Liverani, A.: A Voxel-Based 2.5D Panel Method for Fluid-Dynamics Simulations, in Design Tools and Methods in Industrial Engineering II, C. Rizzi, F. Campana, M. Bici, F. Gherardini, T. Ingrassia, and P. Cicconi, Eds. Cham: Springer International Publishing, 2022, 13–26. https://doi.org/10.1007/978-3-030-91234-5_2
[4] Baker, W. E.; Cox, P. A.; Kulesz, J. J.; Strehlow, R. A.; Westine, P. S.: Explosion Hazards and Evaluation. Saint Louis: Elsevier Science, 2014.
[5] Bilal, S.: Source Panel Method applied to Flow around Cylinder.
[6] Carmichael, R.; Erickson, L.: PAN AIR - A higher order panel method for predicting subsonic or supersonic linear potential flows about arbitrary configurations, in 14th Fluid and Plasma Dynamics Conference, Palo Alto, CA, U.S.A., 1981. https://doi.org/10.2514/6.1981-1255
[7] CFD Consultants GmbH, OpenAPAME.
[8] Chen, M.; Kaufman, A.E.; Yagel, R.: Eds., Volume Graphics. London: Springer London, 2000.
[9] Dobrev, I.; Massouh, F.: Investigation of relationship between drag and lift coefficients for a generic car model, in BULTRANS-2014 Proceedings, Sozopol, Bulgaria.
[10] Filkovic, D.; Virag, Z.: Apame 3D Panel Method, University of Zagreb, 2008.
[11] Hoeijmakers, H. W. M.: Panel Methods in Aerodynamics; Some Highlights, in Panel Methods in Fluid Mechanics with Emphasis on Aerodynamics: Proceedings of the Third GAMM-Seminar Kiel, January 16 to 18, 1987, J. Ballmann, R. Eppler, and W. Hackbusch, Eds. Wiesbaden: Vieweg+Teubner Verlag, 1988, 1–34. https://doi.org/10.1007/978-3-663-13997-3_1
[12] Hughes, F. F.: Computer graphics: principles and practice, Third edition. Upper Saddle River, New Jersey: Addison-Wesley, 2014.
[13] Katz, J.: Race car aerodynamics: designing for speed. Cambridge, MA, USA: R. Bentley, 1995.
[14] Katz, J.; Plotkin, A.: Low-speed aerodynamics: from wing theory to panel methods, New York: McGraw-Hill, 1991.
[15] Kobayashi, K.; Shimizu, H.; Kaiho, M.: Numerical Simulation of Unsteady Turbulent Flow by Voxel Method, JSME Int. J., Ser. B, 47(3), 2004, 477–482. https://doi.org/10.1299/jsmeb.47.477
[16] Krajnović, S.; aDavidson, L.: Flow Around a Simplified Car, Part 1: Large Eddy Simulation, Journal of Fluids Engineering, 127(5), 2005, 907–918. https://doi.org/10.1115/1.1989371
[17] Mohammadikalakoo, B.; Schito, P.; Mani, M.: Passive flow control on Ahmed body by rear linking tunnels, Journal of Wind Engineering and Industrial Aerodynamics, 205, 2020, 104330.
[18] Morse, P. M.; Feshbach, H.: Methods of theoretical physics, New York St Louis San Francisco, McGraw-Hill, 1953.
[19] Onishi, K.; Bale, R.; Tsubokura, M.: Assessment of Rotating Wheel Vehicle Aerodynamics Simulation Using Cartesian Grid Method and Open-Grill Full Vehicle Models, presented at the WCX SAE World Congress Experience, 2019, 2019-01–0660. https://doi.org/10.4271/2019-01-0660
[20] Patil, S.; Ravi, B.: Voxel-based Representation, Display and Thickness Analysis of Intricate Shapes, in Ninth International Conference on Computer Aided Design and Computer Graphics (CAD-CG'05), Hong Kong, China, 2005, pp. 415–422.
[21] Prandtl, L.: 7. Bericht über Untersuchungen zur ausgebildeten Turbulenz, Z. Angew. Math. Mech., 5(2), 1925, 136–139. https://doi.org/10.1002/zamm.19250050212
[22] Schlichting H.; Gersten, K.; Boundary-Layer Theory. Berlin, Heidelberg: Springer Berlin Heidelberg, 2017.
[23] VBBV, 2D panel method for Cp distribution on airfoils.
[24] Vega, G.; Paz, R.; Gleadall, A.; Monzón, M.; Alemán-Domínguez, M. E.: Comparison of CAD and Voxel-Based Modelling Methodologies for the Mechanical Simulation of Extrusion-Based 3D Printed Scaffolds, Materials, 14(19), 2021, 5670. https://doi.org/10.3390/ma14195670
[25] GE jet engine bracket challenge, [Online]. Available: https://grabcad.com/challenges/ge-jet-engine-bracket-challenge. [Accessed: 23-Apr-2020].