

Aerodynamic analysis of the low-drag vehicle concepts with the use of CFD software.

Inż. Krystian Sitko¹, Inż. Piotr Gajec²

¹Cracow University of Technology, Faculty of Mechanical Engineering, Mechanical Department, Institute of Thermal and Process Engineering, al. Jana Pawła II 37, 31-864 Cracow, Poland

²Cracow University of Technology, Faculty of Mechanical Engineering, Mechanical Department, Institute of Thermal and Process Engineering, al. Jana Pawła II 37, 31-864 Cracow, Poland

Abstract

The purpose of this work is the numerical analysis of the vehicle shape influence on the air flow and drag coefficient. Authors proposed four car concepts which must meet special dimension criterions. In the structure must be provided place for a driver seat, steering system and all mechanical components. The all models was designed in 3D program Autodesk Inventor. The paper present localization of the components and elements in the vehicle. Authors defined the boundary conditions to get the streamlined parameters. In the paper also the calculation methods are presented and described. CFD analysis are carried out for proposed geometries. The simulations are performed with the use of the ANSYS Fluent software. In succession was created comparative analysis. It was used to choose the optimal shape. Paper presents useful application of the CFD calculations for the preliminary elimination of inadequate shapes.

Keywords: CFD; turbulence models; numerical modelling; drag force; aerodynamic comparison

1. Introduction

Numerical fluid dynamics (computational fluid dynamics) is a tool used to solve equations describing fluid flow using numerical methods. The use of this type of tools gives incredible opportunities for analyzing flow phenomena, allows them to better understand and thus optimize existing solutions. At the stage of creating new products for many years, it is used as a tool to shorten the time of product development, allowing to stay ahead of competition both temporarily and at the level of quality of the proposed solutions. To take advantage of the existing possibilities, professional software is needed: providing accurate results in a short time.

2. Analysis systems

Ansys software allows simulations to be used to analyze components, assemblies or systems to determine their strength, flexibility, temperature distribution, fluid flow, etc. [1]

1.1. Fluid Flow (Fluent)

Fluent software allows for fluid flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. You specify the computational models, materials, boundary conditions, and solution parameters in Fluent, where the cases are solved.

You can use a Fluent fluid flow analysis system to apply a computational mesh to a geometry within Workbench, then use Fluent to define pertinent mathematical models (for example, low-speed, high-speed, laminar,

turbulent, and so on), select materials, define boundary conditions, and specify solution controls that best represent the problem to be solved. Fluent solves the mathematical equations, and the results of the simulation can be displayed in CFD-Post for further analysis (for example, contours, vectors, and so on).

1.1.1. Range of use

Ansys Fluent allows modeling of laminar and turbulent flows. It also has the latest hybrid models. It enables simulation of multiphase flows, with heat exchange, phase transitions, moving parts and combustion. In the case of modeling very specific phenomena, it allows the addition of new models. [1]

1.2. Fluid flow (CFX)

Ansys CFX enables you to perform fluid-flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. This software allows to import the geometry and meshes, specify the materials, boundary conditions, and solution parameters, solve the calculations, view the results, then create reports using built-in tools. [1]

1.2.1. Range of use

Ansys CFX is used to calculate rotating machines and rigid multiphase problems, because it can solve coupled volumetric functions. [1]

* Corresponding author: krystian.sitko.1995@gmail.com, piotrgajec@interia.pl

3. Geometry

In the initial assumptions, the size of the elements had to be taken into account. The concept foresees that the vehicle should have a seat and basic steering system. It was assumed that a person with a height of 180cm must fit in the vehicle. In addition, at the rear of the vehicle there must have been space for the engine and other mechanical components. On this basis, the minimum dimensions of the vehicle were made.

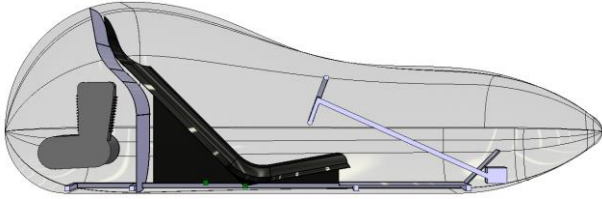


Figure 1. Example of the vehicle with main mechanical components.

Table 1. Boundary conditions of the vehicles.

Boundary conditions	Parameters
Mnimum dimensions of the vehicle BxHxL [m]	0,9 x 0,85 x 2,75
Dimension of the seat and steering system [m]	1,6 x 0,7 x 0,5
Dimension of the engine [m]	0,4 x 0,4 x 0,4

In order to carry out a comparative analysis, four models were designed, which were intended to meet the initial conditions. Each model was supposed to be as streamlined as possible. Geomterries were created as surface models.

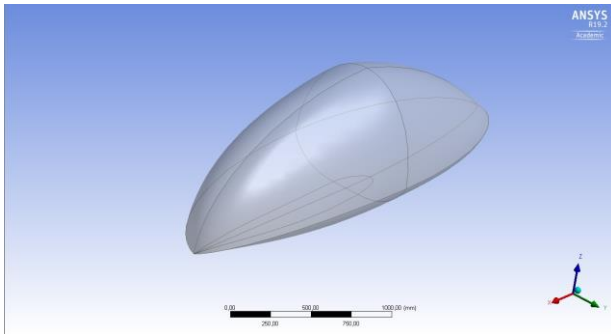


Figure 2. Vehicle geometry nr 1.

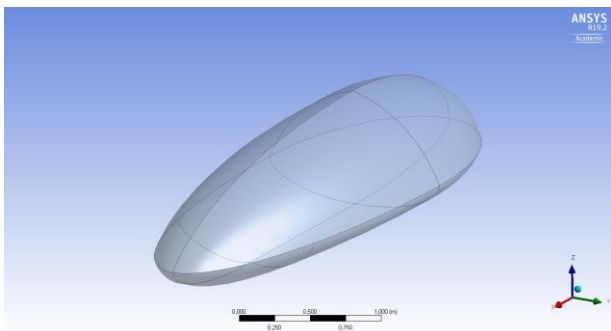


Figure 3. Vehicle geometry nr 2.

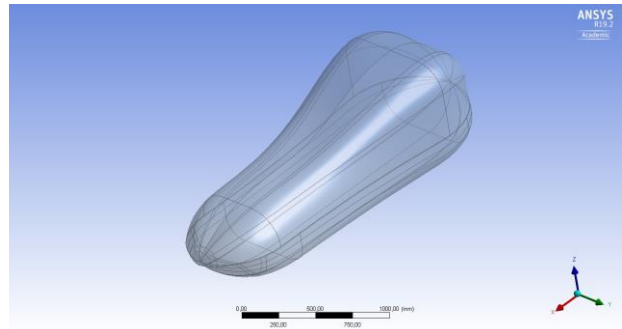


Figure 4. Vehicle geometry nr 3.

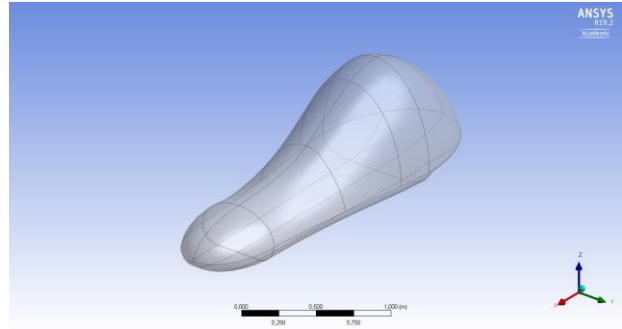


Figure 5. Vehicle geometry nr 4..

3. Boundary conditions for flow calculations

The correctness of performed calculations and their accuracy depend to a large extent on the conditions that are assumed when entering data into the program. Incorrect introduction of even one data can lead to an incorrect result, or the program will not be able to provide a solution. This chapter describes the boundary conditions used by the authors of the article.

3.1. Discretization

Object-based meshing is the recommended meshing approach with which you can generate a tetrahedral, hex-core, or polyhedral volume mesh, with or without inflation layers. In this approach, you first create a conformal, connected surface mesh on all the objects to be meshed. The surface mesh, and material points if required, are then used to identify the regions to be filled with the volume mesh.

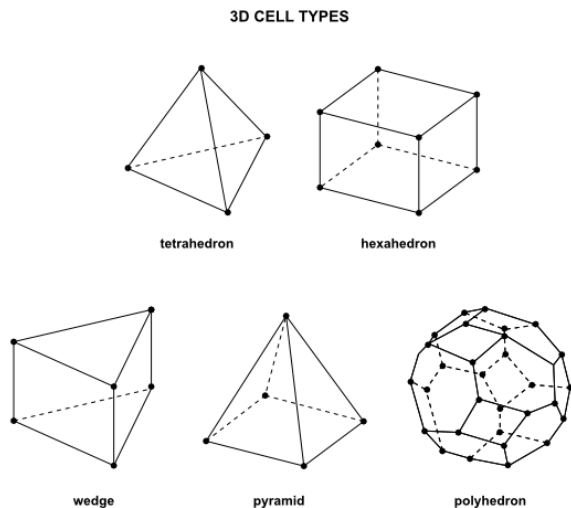


Figure 6. Some examples of meshes that are valid for ANSYS Fluent.[1]

3.1.1. Appropriate Mesh Type

ANSYS Fluent can use meshes composed of tetrahedral, hexahedral, polyhedral, pyramid, or wedge cells (or a combination of these) in 3D. The choice of which mesh type to use will depend on application. When choosing mesh type, consider the following issues:

- setup time
- computational expense
- numerical diffusion [1]

To create a mesh assumed boundary conditions:

Table 2. Boundary conditions of the discretization..

Boundary conditions	Parameters
Element size [mm]	50
Element order	tetrahedron
Physics preference	CFD
Solver preference	fluent

3.2. Flow calculations

Turbulence is the three-dimensional unsteady random motion observed in fluids at moderate to high Reynolds numbers. As technical flows are typically based on fluids of low viscosity, almost all technical flows are turbulent. Many quantities of technical interest depend on turbulence, including:

- Mixing of momentum, energy and species
- Heat transfer
- Pressure losses and efficiency
- Forces on aerodynamic bodies

3.2.1. Modeling Turbulence

Two-equation models are historically the most widely used turbulence models in industrial CFD. They solve two transport equations and model the Reynolds Stresses using the Eddy Viscosity approach. The standard k-epsilon model in ANSYS Fluent falls within this

class of models and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations.

The draw-back of some k-epsilon models is their insensitivity to adverse pressure gradients and boundary layer separation. They typically predict a delayed and reduced separation relative to observations. This can result in overly optimistic design evaluations for flows that separate from smooth surfaces (for example, aerodynamic bodies, diffusers). The k-epsilon model is therefore not widely used in external aerodynamics. [1]

Ansys software provides the following choices of turbulence models:

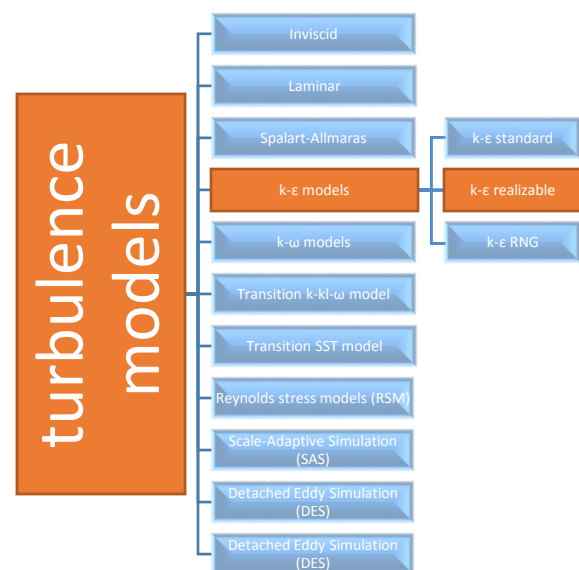


Figure 7. Types of turbulence models.

k-epsilon realizable are used to:

- a) Viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects)
- b) Inclusion of buoyancy effects on epsilon (see Effects of Buoyancy on Turbulence in the k-epsilon Models in the Theory Guide)
- c) Inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models)
- d) Inclusion of compressibility effects (Including the Compressibility Effects Option)
- f) Inclusion of production limiters (Including Production Limiters for Two-Equation Models)

4. Parameters of the optimal vehicle selection

The choice of the most optimal model is associated with the selection of certain parameters that will have a very important role during exploitation. This article

presents the results of the carried out aerodynamic analysis, therefore, in this chapter the parameters were used during the selection of geometry were defined.

4.1. Drag force

A drag force is the resistance force caused by the motion of a body through a fluid, such as water or air. A drag force acts opposite to the direction of the oncoming flow velocity. This is the relative velocity between the body and the fluid.

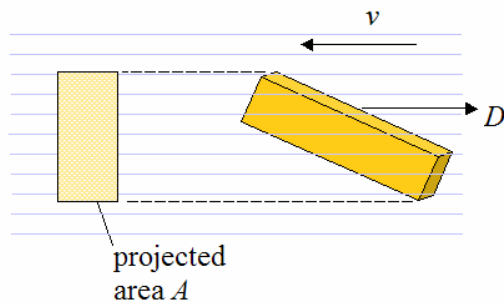


Figure 8. Diagram of drag force. [3]

The drag force D exerted on a body traveling through a fluid is given by formula [3]:

$$D = \frac{1}{2} \cdot C \cdot \rho \cdot A \cdot v^2 \quad (1)$$

where:

C -is the drag coefficient, which can vary along with the speed of the body. But typical values range from 0.4 to 1.0 for different fluids (such as air and water) .

ρ - is the density of the fluid through which the body is moving.

v - is the speed of the body relative to the fluid

A - is the projected cross-sectional area of the body perpendicular to the flow direction (that is, perpendicular to v). This is illustrated in the figure 8.

4.2. Turbulence Intensity

Turbulence Intensity is the ratio of the magnitude of the root-mean-squar turbulent fluctuations to the reference velocity:

$$I = \frac{A \sqrt{\frac{2}{3} k}}{v_{ref}} \quad (2)$$

where:

k -is the turbulence kinetic energy

v_{ref} - is the reference velocity

The reference value specified should be the mean velocity magnitude for the flow. [1]

4.3. Dynamic pressure

Dynamic pressure is the increase in a moving fluid's pressure over its static value due to motion.

In incompressible fluid dynamics, it is indicated as q or Q , defined by:

$$q = \frac{\rho \cdot u^2}{2} \quad (3)$$

where:

q – dynamic pressure in [Pa],

ρ – fluid density in [kg/m³],

u – flow speed in [m/s]. [1]

4.4. Turbulence kinetic energy

In fluid dynamics, turbulence kinetic energy (TKE) is the mean kinetic energy per unit mass associated with eddies in turbulent flow. Physically, the turbulence kinetic energy is characterised by measured root-mean-square (RMS) velocity fluctuations.

In Reynolds-averaged Navier Stokes equations, the turbulence kinetic energy can be calculated based on the closure method, i.e. a turbulence model. Generally, the TKE can be quantified by the mean of the turbulence normal stresses[2]:

$$k = \frac{1}{2} ((u')^2 + (v')^2 + (w')^2) \quad (4)$$

4.5. Static pressure

Static pressure - pressure equal to the value of the force acting on the surface unit, with which two contacting elements of the flowing or resting fluid that are in the given point in the space act on each other.

$$p_s = p_o - \frac{1}{2} \rho v^2 \quad (5)$$

where:

p_o – total pressure of the fluid,

ρ – fluid density,

v – fluid velocity. [1]

5. Results

The following figures (9-17) show a comparison of the results obtained during the analysis. The figures represent places where a particular parameter reaches the highest value and differences that occur between the vehicles. Presenting the results graphically allows to easily read places that negatively affect geometry. All results show maximum value of parameters on the surface.

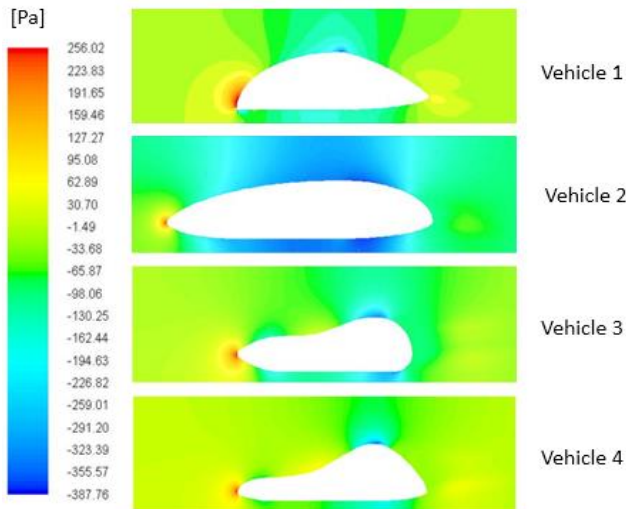


Figure 9. Distribution of static pressure.

In the case of static pressure, it can be seen that the greatest concentration of this parameter occurs on the forehead of vehicles. Its value depends on the tightening of the front of the vehicle. These results can be used to design air ducts to the cockpit.

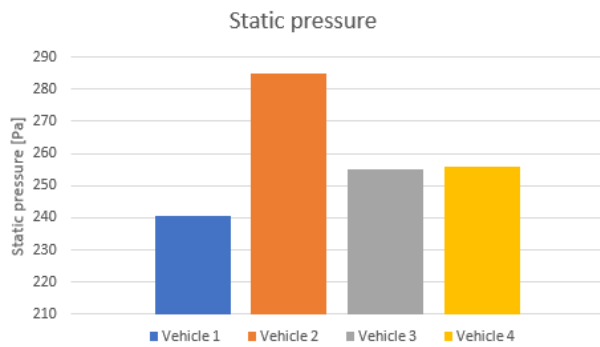


Figure 10. Bar graph of analysed static pressure.

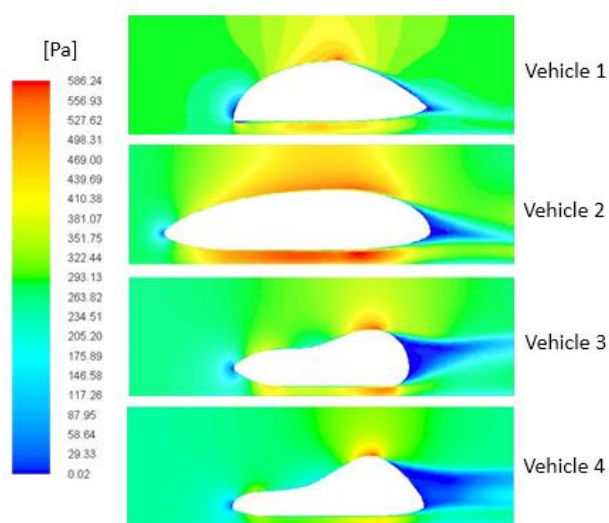


Figure 11. Distribution of dynamic pressure.

Analyzing the dynamic pressure distributions it can be seen that its value increases depending on the air velocity which is dependent on the shape. It can be seen that the

largest ranges of dynamic pressure on the longitudinal surfaces have been created at vehicle 2, which can be caused by a large component of the drag force from the friction between the surface of the vehicle and the air molecules.

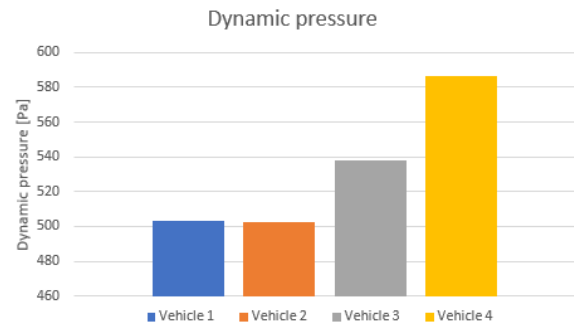


Figure 12. Bar graph of analysed dynamic pressure.

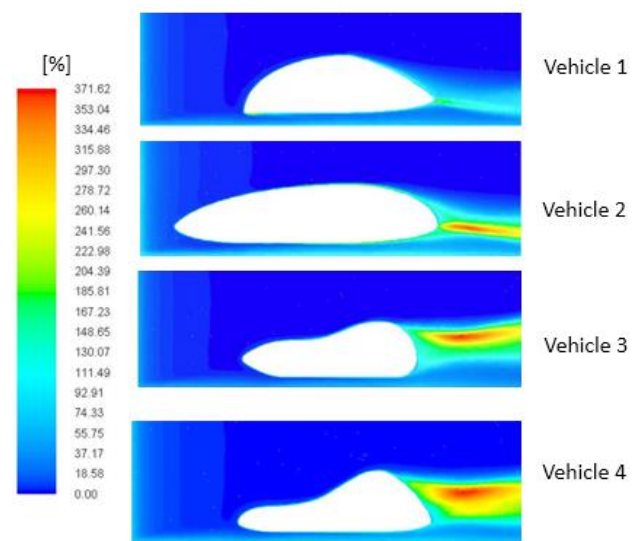


Figure 13. Turbulence intensity.

The intensity of turbulence is caused by the inequality of the object or the intensity of the change of shape that is flown by the air. The more rapid the change in the flow path along the geometry, the greater the intensity of turbulence. The best results have been obtained for the geometry of the vehicle 1, which has a shape similar to a drop of water.

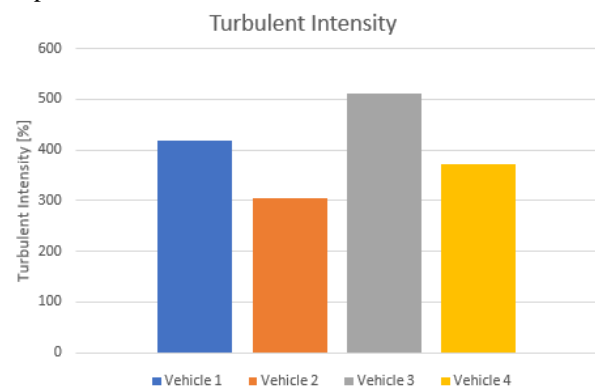


Figure 14. Bar graph of analysed turbulence intensity.

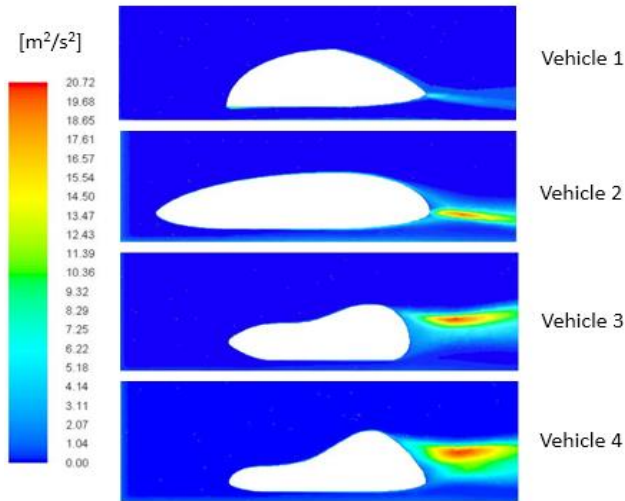


Figure 15. Turbulent kinetic energy.

The kinetic energy of turbulence shows the distribution and size of vortices of the flow stream depending on the shape lines of the vehicle's ending. The more sharp the finish, the greater the turbulence that greatly affects vehicle aerodynamics and resistance.

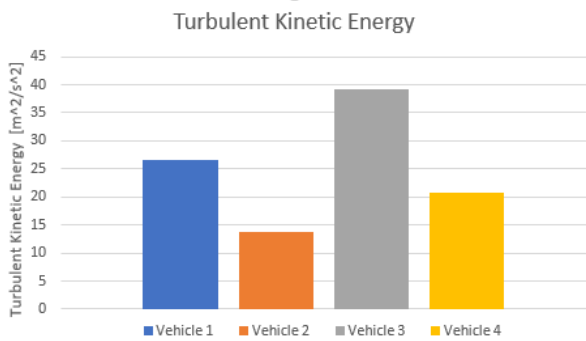


Figure 16. Bar graph of analysed turbulent kinetic energy.

The first vehicle has turbulences in a very small area at the front of the vehicle. They illustrate this figure 14 and figure 16 because the results represent maximum values.

Table 3. Parameters of the vehicles.

	Drag force [N]	Turbulent Kinetic Energy [m ² /s ²]	Turbulent Intensity [%]	Pressure [Pa]		Area [m ²]	number of finite elements
				static	dynamic		
Vehicle 1	15,84	26,61	419,17	240,63	503,54	6,0532	1331686
Vehicle 2	32,85	13,85	303,88	284,69	502,37	10,866	12310547
Vehicle 3	39,71	39,28	511,67	255,06	537,85	6,171	1422846
Vehicle 4	21,26	20,72	371,62	256,02	586,24	5,59	1409279

The resistance force is the basic parameter illustrating the difficulty of moving the object in the fluid. The drag force is proportional to some function of the velocity of the object in that fluid. The model similar to the shape of a drop of water is characterized by the smallest value of the resistance force.

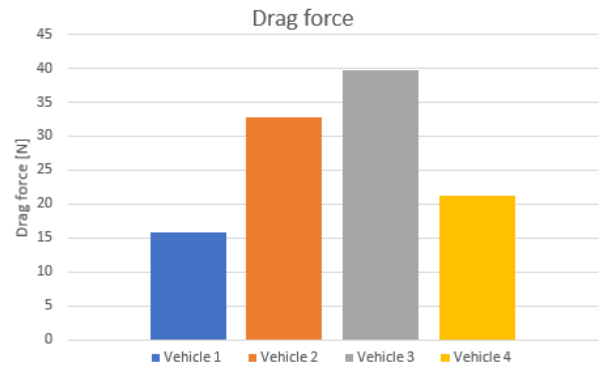


Figure 17. Bar graph of analysed drag force.

6. Comparative analysis

In order to select the most optimal geometry, a construction criterion was prepared. The parameters from the table 3 were used to prepare the table 4.

To create a comparative analysis of selected parameters, a specific coefficient was assigned that shows the hierarchy of parameter impact on model's aerodynamics.

The coefficients have been selected by the authors. Each parameter was assigned the expected value. On this basis, scores were made according to the formula:

$$\text{rating} = \text{coefficient} \cdot \frac{\text{expect value}}{\text{obtained value}} \cdot 100 \quad (6)$$

From the obtained results, the weighted average was calculated according to the formula:

$$\text{result} = \frac{\text{sum of ratings}}{\text{sum of coefficients}} \quad (7)$$

The best vehicle has the highest value of the weighted average.

Table 4. Results of the comparative analysis.

	Drag force	Turbulent Kinetic Energy	Turbulent Intensity	Pressure		Area	Value of of coonstruction criterion
				Static	Dynamic		
expected value	15	10	300	200	400	5	-
coefficient	0,8	0,6	0,5	0,2	0,4	0,1	-
Vehicle 1	75,7576	22,5479	35,7850	16,6230	31,7750	8,2601	73,3649
Vehicle 2	36,5297	43,3213	49,3616	14,0504	31,8490	4,6015	69,1206
Vehicle 3	30,2191	15,2749	29,3158	15,6826	29,7481	8,1024	49,3626
Vehicle 4	56,4440	28,9575	40,3638	15,6238	27,2926	8,9445	68,3178

7. Conclusion

The article presents the method of using aerodynamic analysis to choose the optimal geometry. For this purpose, 4 vehicle models were designed. A calculation was made for the assumed boundary conditions. On the basis of the results obtained, a construction criterion was prepared, which allowed to choose the optimal shape. Computa-

tional fluid dynamics are used to carry out flow simulations. At the stage of creating a product, these types of tools are used to shorten the time of development of a new product, to check prototypes, which reduces the costs and time of the production process.

References

- [1] *Ansys Fluent 19.2, Ansys Workbench Help*
- [2] *www.fwmt.put.poznan.pl [opory podczas opływania ciał]*
- [3] *<https://www.real-world-physics-problems.com/drag-force.html>*