

Numerical Analysis of The Profile in the Aero Tunnel

Ivana Terzić ^{1*}, Slobodan Aleksandrov ¹, Milica Todorović ¹

¹ College of Applied Mechanical and Technical Engineering, Trstenik, Serbia

* ivana.terzic@vtmsts.edu.rs

Abstract: Contemporary product and process development involves the application of various software tools and technologies. The assessment of aerodynamic characteristics is an essential part of the optimal aero profile design process. This paper presents a numerical analysis of the aero tunnel profiles using Catia and Ansys Fluent software packages, and obtaining an effective method of determining the aerodynamic characteristics of the profile, which is based on viscous-non-viscous interaction. Ansys Fluent software package provides a wide range of tools for efficient application in the design of aero profiles, which reduces the time and cost of developing aero profiles compared to the classic approach to the development of physical models. The paper presents the graphs of the obtained values of the field velocities and pressures of the tested profile in a modeled aero tunnel with given boundary conditions. Special attention is devoted to monitoring the pressure changes and the air flow velocity at an object in the aero tunnel.

Keywords: aerodynamic; aero tunnel; modeling; numerical analysis

1. INTRODUCTION

A computer design system is a set of software tools and technologies that provide engineers with a number of benefits while working on the development of product design. Computer modeling is based on geometric transformation. Continuing market demands, as well as market competition for new and cheaper products, have a direct impact on the faster development of computerized CAD (Computer Added Design) programs. On the other hand, CFD (Computational Fluid Dynamics) presents a branch of fluid mechanics in which numerical analysis and data structures are used to solve and analysis fluid flow problems. Modern three-dimensional (3D) software packages are used for modeling and simulating the interaction of liquids and gases with surfaces defined by boundary conditions.

In this paper, a 3D analysis of the aerodynamics of the physical model in an aero tunnel is presented using a software package for numerical analysis of fluid dynamics. The simulation of the air flow in the aero tunnel and the aerodynamic characteristics of the given profile were performed. The procedure for 3D modeling of the object and aero tunnel is presented using the CATIA V5 software package and simulation of fluid flow through the aero tunnel, using the Ansys Fluent software package.

2. 3D MODELING AND CATIA V5 SOFTWARE PACKAGE

Nowadays modeling cannot be imagined without the use of modern CAD systems, which enable both 2D and 3D modeling of parts, production of circuits, generation of technical documentation, simulation of work and analysis of elements and systems. Previously developed software programs for 3D modeling can be divided into:

- Software programs for drawing contours and diagrams;
- Software programs for shell modeling;
- Software programs for modeling the shape of geometric bodies, machine parts and assemblies.

In the paper, the CATIA V5 software and its Part Design module, as well as Sketcher drawing module (Fig. 1), were used to create 3D models.

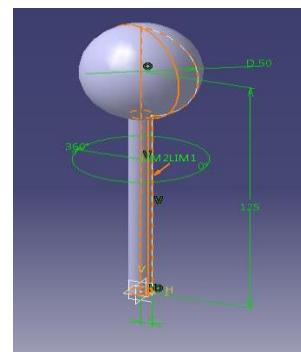


Figure 1. The tested profile modeled in the CATIA V5 software package

Fig. 2 shows the cross section of the formed aero tunnel with the profile set.

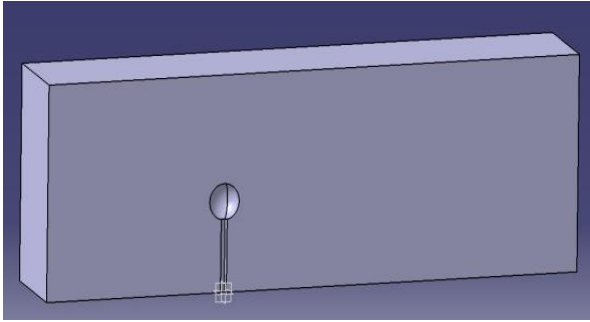


Figure 2. Cross section of the aero tunnel with profile

Volumetric modeling refers to the process of creating parts by combining volumetric objects to obtain a 3D model. Such models are based on 2D sketched profiles. Finished technical elements of type draft, fillet, chamfer or thickness are created by commanding the command over the existing geometry of the 3D model to create a more complex model, i.e. in order to carry out its upgrade, which is explained in more detail in [1].

CATIA V5 is a software program used in the automotive and aerospace industry. It represents an associative system that includes and connects all the stages and activities of the production company. The program's association and flexibility enable the implemented changes in the model to be reflected in all stages of product development, which increases the control of the implemented changes on the model and therefore becomes available to all members of the product development team.

Since these are very demanding graphical tools, it is necessary to use multi-core processors in order to achieve the proper process speed when creating virtual 3D models [1, 2, 3, 4].

2.1. Simulation procedure and network generation

Modeling of the product prior to its use in order to meet the requirements for better, better quality, reliable and durable characteristics can be performed by simulations that evaluate the behavior and design of new products.

The Ansys software program contains CFD analysis that includes analytical areas, from aerodynamics, combustion, heat transfer, chemical reactions and many other areas where problems can be solved by numerical simulations of fluid mechanics. Ansys Fluent enables complete mesh flexibility, including the ability to solve flow problems using unstructured meshes that can be relatively easily developed into complex geometries. Ansys Fluent is a software program that is ideal for simulating compressible and

unstable fluids in complex geometries, including the use of alternative physical methods [5, 6].

After starting the ANSYS Workbench module, a start window opens where basic numerical simulations are selected in order to load the geometry of the aero tunnel from the CATIA software program. The next step is to start the ANSYS Meshing, where there are options for setting global mesh parameters, in this case the defaults-CFD and the Solver-Fluent list (Fig. 3).

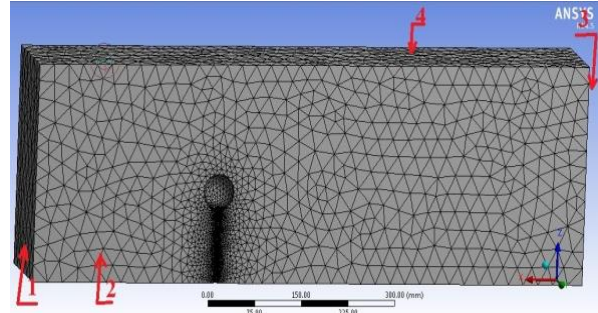


Figure 3. Generated mesh by the Tetrahedrons method with given boundary conditions

The model with the given boundary conditions is ready for numerical analysis in ANSYS Fluent.

The resulting network is the result of defined global parameters. Changing these parameters will result in a different meshing. In complex geometries, it is very difficult to obtain a high-quality mesh; therefore, the block-based approach to mesh forming provides the possibility of obtaining a better quality mesh by defining local parameters. In order to gain the best possible control of the mesh, it is necessary for each volume to define the method of generating the mesh [7].

3. STARTING ANSYS FLUENT MODULE

After the ANSYS Fluent module is started, the initial window opens in which basic numerical simulations are selected in order to load the generated mesh of the test aero tunnel, the File>Read>Mesh command (Fig. 4). After loading the mesh, there is a message about the number of nodes where it should be noted that there is no warning in the comments, as this is a sure indicator that the mesh has not been correctly generated.

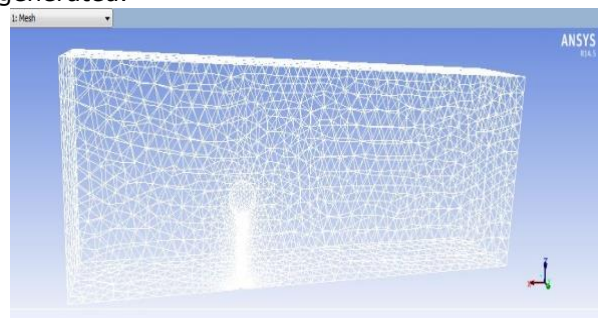


Figure 4. Loaded generated mesh

The definition of a mathematical model should be done according to the type of fluid flow. The k-omega (2 eqn) option is selected from the Model list and after that the corresponding fluid is assigned in this case it is air.

The boundary conditions set the parameters at the input and exit from the domain. At the input, the air flow velocity is set at 10 m/s; in the same way, set the specified values at the exit from the domain, while the boundary conditions on solid surfaces of the estimated domain are automatically assigned (Fig. 5) [8, 9].

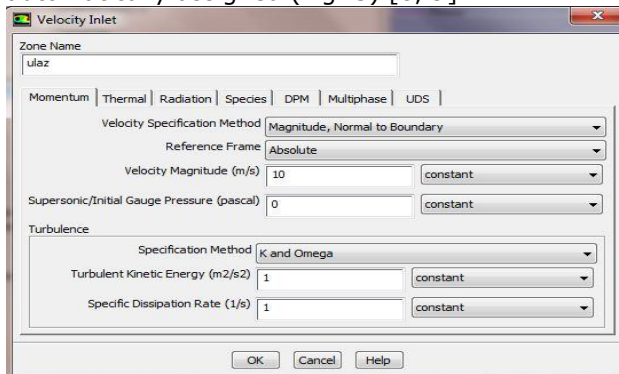


Figure 5. Setting boundary conditions

3.1. Setting up the initial solution-initializing the iterative process

In order to complete the simulation of fluid flow through the aero tunnel, it is necessary to set appropriate parameters through which the characteristics of the fluid flow can be controlled. By setting parameters for iterative monitoring, defining a surface monitor at the output, by setting up initial solutions, i.e. initialization comes to the start of the iterative process (Fig. 6).

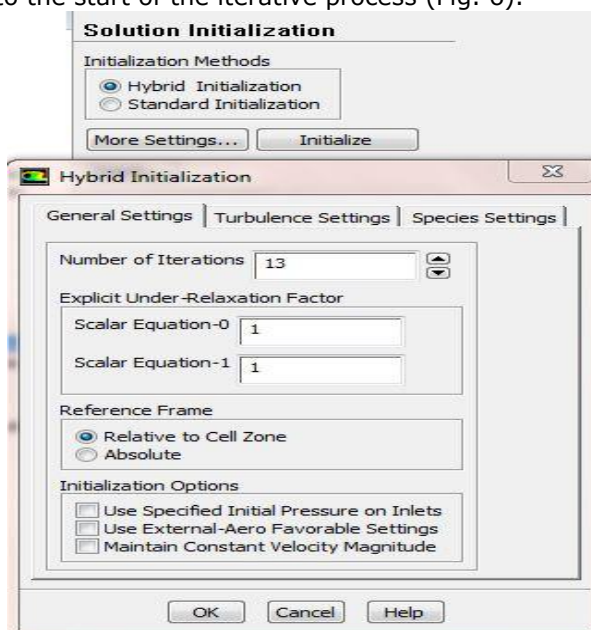


Figure 6. Initialization of the iterative process

Through the Run Calculation card, the number of the desired iteration is assigned, in this case 1000

iterations are selected, which gives a graphical representation of the flow of the iterative process (Fig. 7).

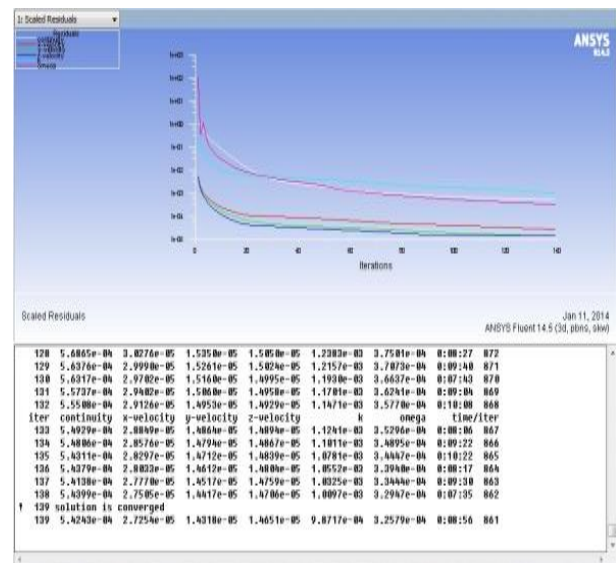


Figure 7. Graphical representation of the iterative process

In Fig. 7 it is noticeable that the process converged on the 127th iteration for the entire model and the process lasted 2 minutes and 38 seconds. The process was done for half of the model where the solution converged on the 139th iteration, but the iteration process lasted 1 minute and 28 seconds.

The convergence process will stop when residuals reach a certain value or after a given number of iterations, in this case it is 1000 iterations. The exact number of iterations varies depending on which platform is being used, and a dialog with information is given that tells us about the convergence [10].

The point at which solutions are no longer changing with ever subsequent iteration is convergence. Convergence criteria, together with the reduction of residuals, also help in determining the complete simulation solution. Convergence criteria represent predefined conditions over residuals that indicate a certain level of convergence achieved [11].

3.2. Creating a level of symmetry for a full model tested in an aerotunnel

The free flow of fluid in the aero tunnel was obtained by the numerical domain, where the size of the domain analysis was made for the full profile of the aero tunnel in accordance with the recommendations [12]. In solving the problem, the analysis was compared to the plane of symmetry, since the software package provides this approach. A symmetrical plane for the full model was formed according to Fig. 8.

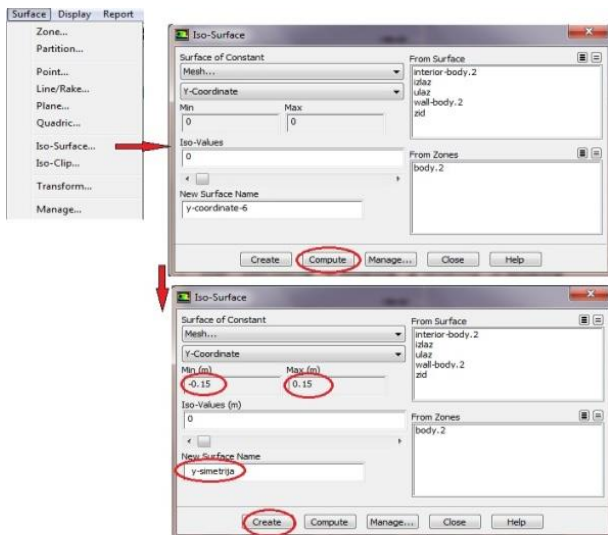


Figure 8. A procedure for forming a symmetric plane for a full model of aeroprofile

The input speed is given as a boundary condition, defined at 10 m/s, on one side of the domain, while on the other side of the domain a pressure limit (0 Pa) is used.

The air density during the analysis was 1.225 kg/m³, dynamic viscosity 1.7965 E-5 kg/(ms) and ambient pressure 101325 Pa.

3.3. Analysis of results

Post-processing of results in ANSYS-it is scored through the Graphics and Animations tab>Contours; Vectors>Set Up (Fig. 9, 10 and 11) to display the speed field, the pressure field, and the velocity vector display.

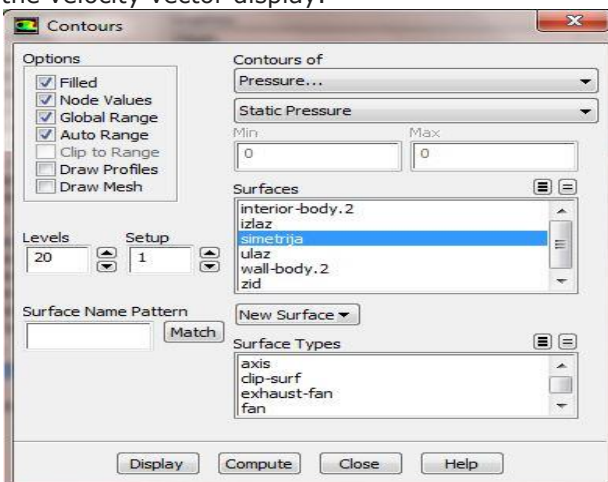


Figure 9. Post-processing of the pressure field

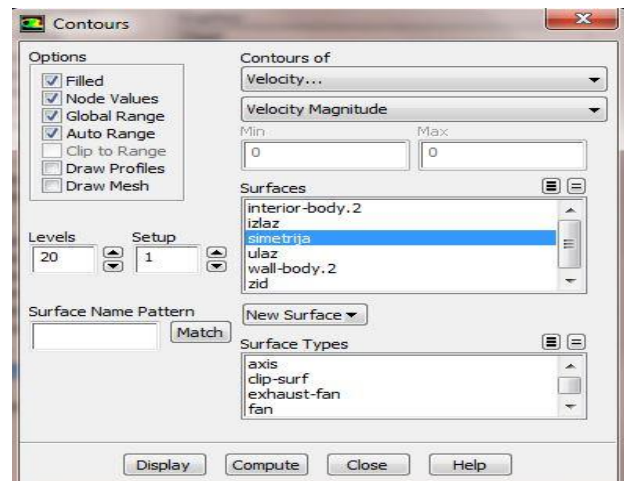


Figure 10. Post-processing of the field of speeds

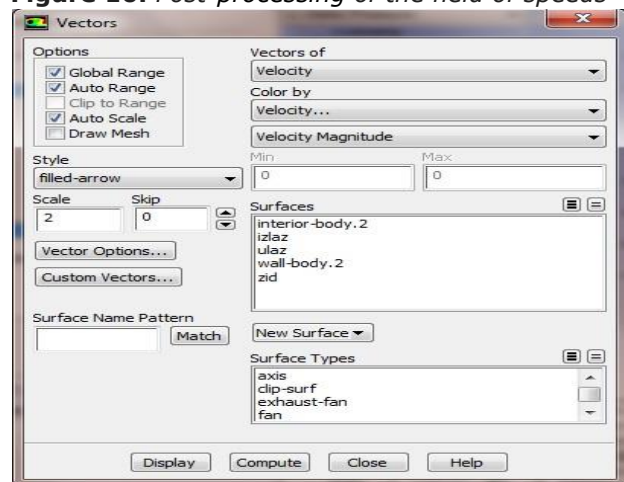


Figure 11. Post-processing of the vector velocity display

Post-processing (analysis of results) gives us an insight into the modeling of the desired process with calculated values.

In the second part of the analysis, the results of simulation of the speed and pressure for the analyzed profile were obtained (Fig. 12, 13 and 14).

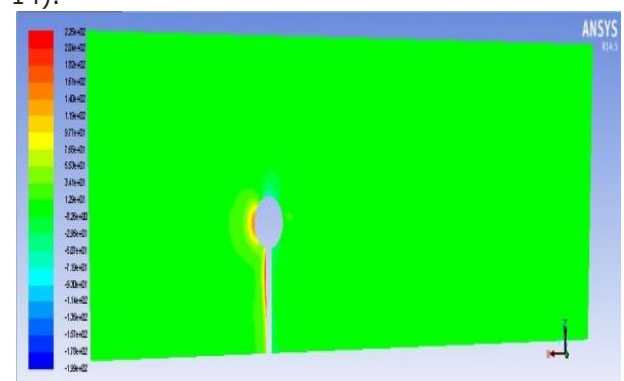


Figure 12. Display of the pressure field for the whole model in the aero tunnel

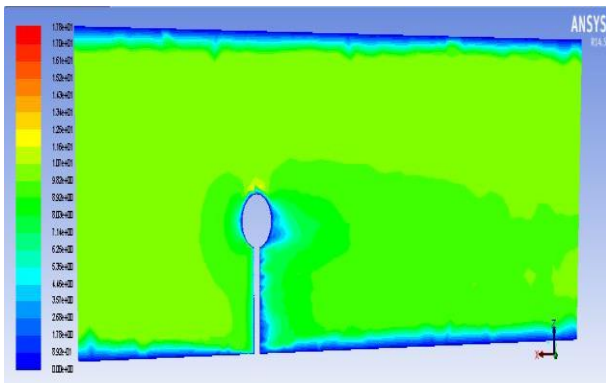


Figure 13. Display the velocity field for the entire model in the aero tunnel

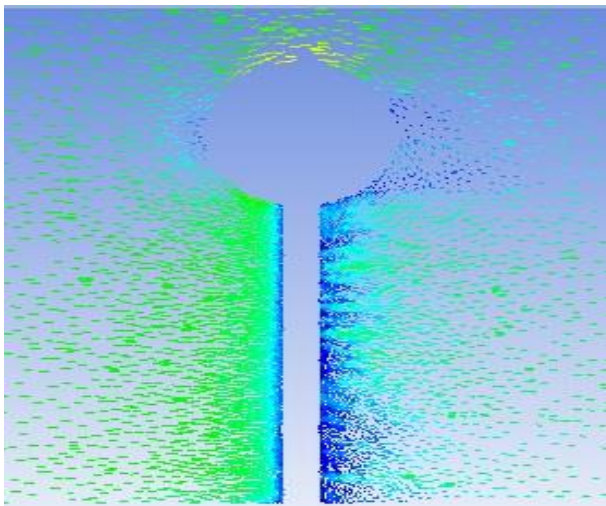


Figure 14. Vector display of speeds for the entire model of the tested profile in the aero tunnel

In figures 12, 13 and 14, the obtained values of the tested model of the profile in the aero tunnel are presented, where the pressure and velocity fields are clearly formed in relation to the given boundary conditions. Obviously, the software provides great opportunities in the profile construction itself to get the proper aerodynamic characteristics of the profile being tested.

By setting all of the steps presented, the conditions for diagramming the flow parameters are created. The Plot card is used for diagrammatic pressure display, and through it in the Options list activates Node Values and Position on X Axis. In the Plot Direction list for box X, number 1 is selected, then for field Y number 0. From the drop-down list Y Axis Function, Pressure was selected and then Static Pressure. From the X Axis Function drop-down list, Direction Vector was selected, and in the Surfaces field, symmetry was selected and finally the Plot command was activated (Fig. 15).

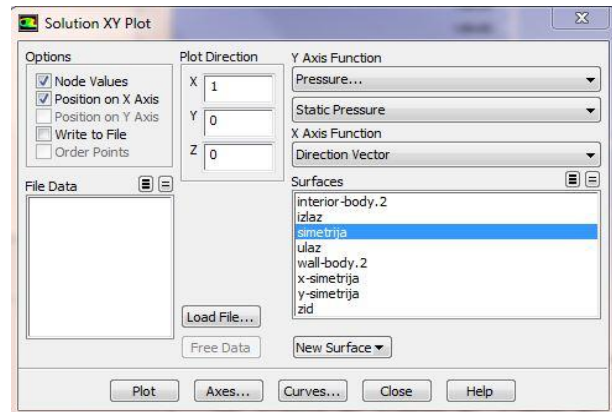


Figure 15. Plot card adjustment procedure for displaying the pressure diagram

After setting the parameters and activating the Plot card, a diagram pressure display is obtained (Fig. 16).

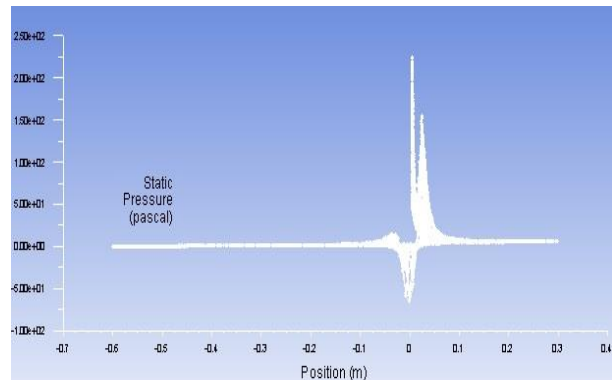


Figure 16. Diagram display of the pressure made for the whole model of the profile examined in the aero tunnel

This way of analysis gives a great advantage in the testing and simulation of the flow of the process itself [13, 14, 15].

4. CONCLUSION

Modeling the process as well as interpreting the results of modeling and simulation help in analysis and setting the conditions for applying to real processes. Numerical simulation provides a detailed analysis of simulation results, so that based on the analysis, corrective measures can be proposed, optimized project solutions as well as identification of process influencing parameters. Optimizing the design solution through numerical simulations greatly influences the reduction of the cost of testing and construction.

The results of the simulation of the aero tunnel profile obtained in this paper provide an insight into the characteristics of the movement of the speed parameters and the pressure of the fluid flow in the aero tunnel.

REFERENCES

- [1] Cad Cam Design Centar, „CATIA V5 knjiga II: Napredni tečaj (Part Design, Assembly Design, Drafting, Surface)“, Zagreb, 2003.
- [2] Cad Cam Design Centar, „CATIA V5 knjiga I: Početni tečaj (osnove modeliranja)“, Zagreb, 2003.
- [3] Nurdin, Ć.: *Modelling of protective casing, decorative cover, shift lever handler and shift lever of car gerabox*, <https://hrcak.srce.hr/101173> (04.04.2018.).
- [4] Karam, F.; Kleismit, C.: *Catia V5*, Kompjuter biblioteka, 2004.
- [5] http://www.fe.um.si/images/jet/Volume10_Isue1_Paper1.pdf (13.04.2018.).
- [6] <https://www.sharcnet.ca/Software/Fluent6/pdf/gs/flgs.pdf> (13.04.2018.).
- [7] Simon, M.; Matej, Š.: *Numerical analysis of an ahmed body with different software package*, Journal of energy technology, Volume 10 / Issue 1 March 2017.
- [8] <http://www.ctc.kg.ac.rs/clanci/numericke-simulacije-procesa-ulazni-podaci-za-fe-simulaciju.html> (25.02.2018.).
- [9] [http://www.rose-hulman.edu/~luic/ME427/FLUENT/Tutorial1 - Fluid Flow and Heat Transfer in a Mixing Elbow.pdf](http://www.rose-hulman.edu/~luic/ME427/FLUENT/Tutorial1-FluidFlowandHeatTransferinAMixingElbow.pdf) (25.02.2018.).
- [10] Heat Transfer & Fluid Flow Simulation with ANSYS, Keerati Sulaksana; Phattharaphan Thamatkeng; School of Mechanical Engineering Suranaree University of Technology, [http://eng.sut.ac.th/me/2014/document/training/CAETraining\(Fluid\).pdf](http://eng.sut.ac.th/me/2014/document/training/CAETraining(Fluid).pdf) (13.04.2018.).
- [11] <https://www.sharcnet.ca/Software/Fluent6/pdf/gs/flgs.pdf> (04.04.2018.).
- [12] Salleh, M. K. A. B.: *Simulation and analysis drag and lift coefficient between sedan and hatchback car*, Bachelor thesis, PAHANG: University Malaysia Pahang, 2009.
- [13] http://www.cadfamily.com/download-pdf/ANSYS_Rotordynamics/ANSYS.Mechanica.Tutorials.pdf (25.02.2018.).
- [14] <http://www.ctc.kg.ac.rs/clanci/numericke-simulacije-procesa-ulazni-podaci-za-fe-simulaciju.html> (25.02.2018.).
- [15] [http://www.rose-hulman.edu/~luic/ME427/FLUENT/Tutorial1 - Fluid Flow and Heat Transfer in a Mixing Elbow.pdf](http://www.rose-hulman.edu/~luic/ME427/FLUENT/Tutorial1-FluidFlowandHeatTransferinAMixingElbow.pdf) (25.02.2018.).