Journal of Energy Technology

JET Volume 10 (2017) p.p. 11-21
Issue 1, March 2017
Type of article 1.01
www.fe.um.si/en/jet.html

NUMERICAL ANALYSIS OF AN AHMED BODY WITH DIFFERENT SOFTWARE PACKAGES

NUMERIČNA ANALIZA AHMEDOVEGA TELESA Z RAZLIČNIMA PROGRAMSKIMA PAKETOMA

Simon Muhič[ℜ], Matej Štefanič[1]

Keywords: Computational Fluid Dynamics, Ahmed Body, Numerical analysis, Ansys Fluent, SolidWorks Flow Simulation

Abstract

In this article, the results of CFD simulations are compared using two different software packages for numerical fluid dynamics. The analysis is performed for an Ahmed body, for which the measurement results and a variety of numerical simulations are available in the literature. The results of the stationary CFD simulations with the RANS approach show a significant difference between the results obtained with the SolidWorks Flow Simulation 2014 software and ANSYS Fluent 16.2 software in the air flow analysis area from 10 m/s to 60 m/s. The difference in computational time is also apparent.

Povzetek

V članku primerjamo rezultate CFD simulacij, pridobljene s pomočjo dveh različnih programskih paketov za numerično dinamiko tekočin. Analiza je izvedena za Ahmedovo telo, za katerega so v literaturi na voljo tako rezultati simulacij, kakor tudi meritev. Rezultati CFD simulacij kažejo na signifikantno razliko med rezultati, ki smo jih pridobili s programsko opremo SolidWorks Flow Simulation 2015 in s programsko opremo ANSYS Fluent 16.2 v analiziranem območju toka zraka od 10 m/s do 60 m/s.

ℜ Corresponding author: Simon Muhič, Faculty of Technologies and Systems, Na Loko 2, SI-8000 Novo mesto, Slovenia, Tel.: +386 7 393 019, E-mail address: simon.muhic@fts-nm.si

[1] Faculty of Technologies and Systems, Na Loko 2, SI-8000 Novo mesto, Slovenia

1 INTRODUCTION

CAE (Computer Aided Engineering), which refers to the extensive use of computer software to assist in engineering tasks and analyses, is increasingly evolving in engineering practice. It includes software for the analysis of solids using the method of finite element analysis (FEA), followed by software for analyses in the field of numerical fluid dynamics (Computational Fluid Dynamics (CFD)), electromagnetics, as well as coupled physics. The CAE software can also include add-ons to automatically optimize the analysed products. Analyses are very important in the development of products for achieving a better product quality. Engineers perform the analyses in the early development stage, where a virtual prototype of the product is made; after that, it is analysed and, if necessary, optimised to obtain the most optimal product which is sent to the production process. Certainly for virtual engineering, high-performance software that enables such analyses and high-performance computers for the processing of numerical data are both necessary. Such product development makes the development process cheaper and enables the company to be more competitive due to a shorter time for the development to the final product to be sent to the market. CFD analyses cover various analysis areas, from aerodynamics, combustion, heat transfer, chemical reactions, to many other areas where problems can be solved by numerical simulations of fluid mechanics. The software can be independent or, as recently common, in the form of an addition to 3D modellers. Therefore, the question that often arises is what software to choose. Due to the popularity of the high-performance software in the field of CFD, ANSYS Fluent on one side, and the prevalence of the CAD software SolidWorks enabling CFD analyses through its Flow Simulation interface, on the other, we opted for a direct comparison between the two software packages. Several analyses have been made, and the results do not show significant discrepancies between them, *[1, 2]*.

For a direct comparative analysis, we performed an analysis of the Ahmed body, which was tested in the 1980s, *[3]*. The analysis was performed using the ANSYS Fluent and SolidWorks Flow Simulation software. We analysed the prediction accuracy as well as the speed of the software in solving the model. For the basic comparison, we compared the results of drag coefficient and lift coefficient. Figure 1 shows the Ahmed body for which the numerical analyses were performed. The body has a simple form on which the basic aerodynamic characteristics can be established, and several numerical analyses have been performed on it, such as *[4, 5, 6, 7].*

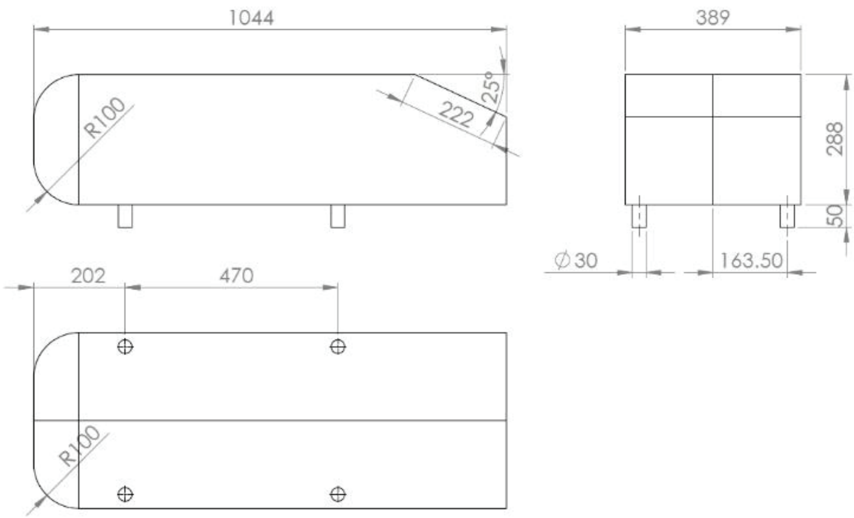

***Figure 1**: Ahmed body*

The Ahmed body drag coefficient C_D in dependence on Reynolds number is given in the equation 1 [4]. The equation is obtained by the statistical analysis of the experimental data.

$$C_D = 0.2788 + 0.0915 \cdot e^{\left(\frac{-Re \cdot 10^{-6}}{1.7971}\right)} \qquad (1.1)$$

2 VEHICLE AERODYNAMICS

Vehicle aerodynamics influences fuel consumption, vehicle stability, aerodynamic drag and the resulting noise. Drag force F_D acts in the opposite direction of the vehicle's movement, in the vehicle's direction. The first part of the force results from aerodynamic drag according to the form of the vehicle, the second part acts as a result of friction between the vehicle's surface and fluid which surrounds the vehicle. Figure 2 shows the action of forces on the vehicle.

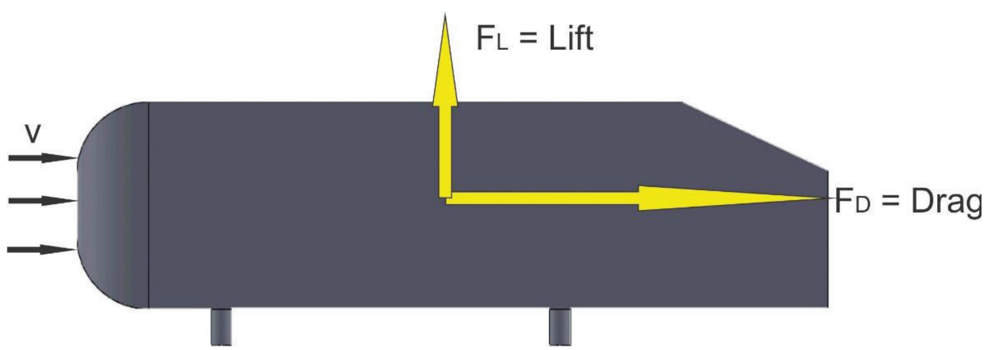

***Figure 2:** Forces on the Ahmed body, [8]*

Drag coefficient C_D is the ratio between the drag force and dynamic force defined by the equation:

$$C_D = \frac{F_D}{p_d A} \tag{2.1}$$

As the surface A we consider the complete front head part, the dynamic pressure p_d being defined by velocity in front of the body. Similarly, lift coefficient C_L is defined with the equation:

$$C_L = \frac{F_L}{p_d A} \tag{2.2}$$

In this case, lift force F_L acts on the vehicle due to the surrounding air flow and the velocity difference between the lower and upper part of the vehicle.

3 NUMERICAL MODEL

For the numerical model, the RANS modelling approach was used. For modelling turbulence in the ANSYS Fluent software, the Realizable k-ε model (RKE) was used. To obtain boundary layers, the Enhanced Wall Treatment wall function was selected; it makes the k-ε model more suitable for treatment in the wider use of the non-dimensional variable y^+. In addition to the aforementioned model, the Non-Equilibrium wall function model, recommended for analysis in aerodynamics, was also used, *[9]*. The demand for the convergence value was that the residuals are less than 0.0005. The COUPLED scheme and the second order solvers were used. Numerical domain in the ANSYS Fluent software was divided into 12 volumes, allowing more types of numerical mesh and a better control of the individual parts of the domain (**Figure 3**).

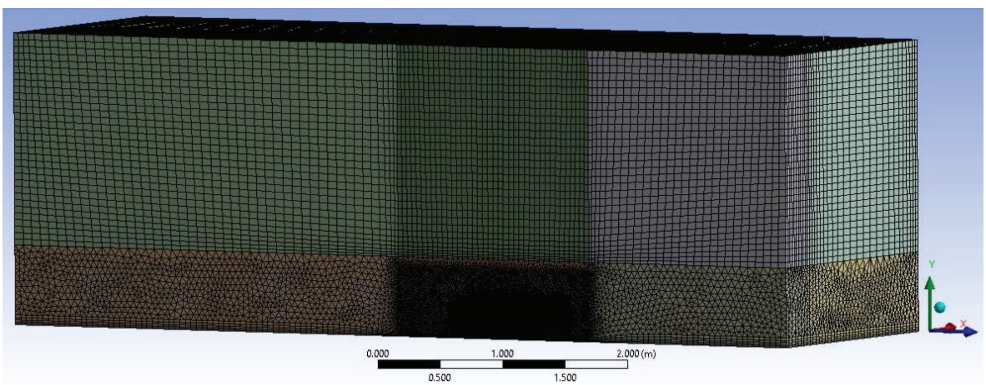

Figure 3: *Numerical mesh in ANSYS Fluent*

In the volumes above the body, a hexahedral mesh was used; around the body, however, a tetrahedral mesh was generated. In doing so, to obtain the boundary layers, we used the Inflation function on the model, allowing us to properly host the mesh on the walls (Figure 4).

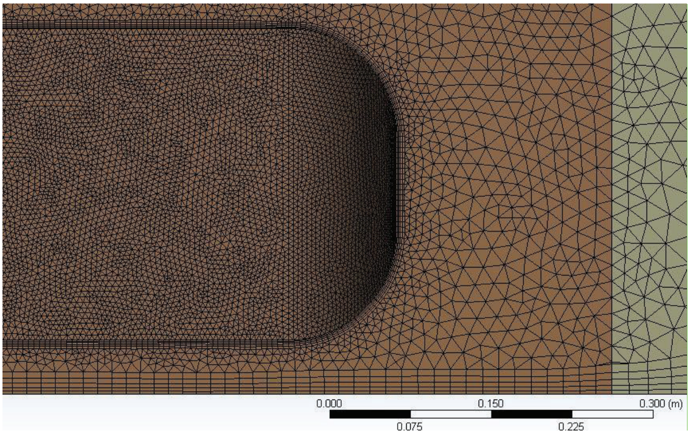

Figure 4: *Detail of numerical mesh near Ahmed body*

In the SolidWorks Flow Simulation software for modelling turbulence, the k-ε turbulence model was also used. For wall function, the Two scale Wall Function was used, which essentially offers a similar solution as the Enhanced Wall Treatment setting in the ANSYS Fluent software. The convergence value was set on automatic. SolidWorks Flow Simulation uses only Cartesian mesh (Figure 5). Using the software tools, an additional meshing of the volume was performed around the body (Local Mesh Sizing). Several analyses were performed using a variety of meshes for finding a solution that is independent of the computational mesh. In addition, an analysis with adaptive mesh technology for the inflow velocity of 40 m/s was performed, the maximum number of cells was limited to 6000000 with a triple level of adaptation. A periodic adaptation of every 50 iterations was also used. For other analyses and velocity between 10 m/s and 60 m/s, the adaptive meshing was turned off.

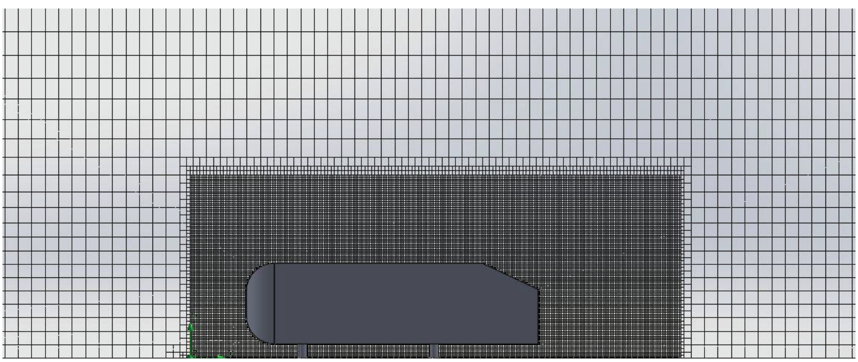

Figure 5: *Numerical mesh in SolidWorks*

4 NUMERICAL DOMAIN AND BOUNDARY CONDITION

The numerical domain was large enough to obtain a free flow, without the impact of domain walls to the geometry analysis. The size of domain analysis was specified for both analyses in accordance with the recommendations, *[8]*. In dealing with the problem, we decided for the analysis with respect to the plane of symmetry, since the software packages allow such approach. The Velocity Inlet was given as a boundary condition, in which we were changing the incoming velocity between 10 and 60 m/s on one side of the domain; on the other side of the domain, we used the Pressure Outlet boundary condition, where a pressure outlet without gauge pressure (0 Pa) was set. Air density during the analysis was 1.225 kg/m^3, dynamic viscosity 1.7965 E-5 kg/(m s), and the surrounding pressure 101325 Pa. The analysis was performed for an incompressible fluid and isothermal conditions.

5 RESULTS

5.1 Referential example

Table 1 shows the results of the numerical analysis at the referential air inflow velocity of 40 m/s and measurements, *[4]*. It can be seen that in the analysis with ANSYS Fluent software using the Realizable k-ε turbulence model with Non-Equilibrium walls treatment (Fluent RKE-NEQ) the aerodynamic drag coefficient deviates from the measured value by -0.1%, the lift coefficient deviates by 1.9%. The variable y+ on the walls of the body in this instance was $y+_{min}$ = 3.5, $y+_{max}$ = 115 in $y+_{ave}$ = 60.

A larger deviation, however, can be found in the results using the software SolidWorks Flow Simulation and adaptive mesh thickening technique (SolidWorks AM), with which we can see that the aerodynamic drag coefficient deviates from the measured value by -9.7%, and the lift coefficient deviates by -19.4%. Drag force, which was in this case calculated with the ANSYS Fluent software, amounts to 16.6 N, and lift force is 19.8 N. Drag force, which was calculated with the CFD software package SolidWorks and adaptive technique, in the final mesh of 6.7 million finite elements, amounts to 14.8 N, and lift force is 12.9 N. The differences in the computational time is crucial. Using the ANSYS Fluent package, the computational part of the analysis took us 43 minutes, simulation performed with a quad-core CPU, and using the SolidWorks software, the time increased to 65 hours.

Table 1: *Comparison of the results at the inflow velocity of 40 m/s*

	Experiment [4]	Fluent RKE-NEQ		SolidWorks	
			Δ%		Δ%
C_D	0.298	0.297	-0.1	0.269	– 9.7
C_L	0.345	0.352	1.9	0.278	– 19.4

5.2 Analysis at different velocities

In the second part of the analysis, the results of simulations for different velocities of the air inflow were compared. Using the ANSYS Fluent software, an analysis of simulations with the Realizable k-ε turbulence model with Non-Equilibrium walls treatment (Fluent RKE-NEQ), as well as with Enhanced Wall Treatment (Fluent RKE-EWT), was performed. Using the SolidWorks Flow Simulation software, the case with two meshes, with Coarse (SW-Coarse) and Fine (SW-Fine) mesh without using the adaptive mesh technique ware analysed; the Coarse mesh had 2.5 million finite elements, and the Fine mesh 6.5 million elements. Figure 6 displays the results of the calculated drag and lift forces in dependence on the inflow velocity within the range of 10 m/s to 60 m/s, and Figure 7 displays the drag and lift coefficients within the analysis area. At first glance, the results of the different models appear similar for each of the numerical meshes and analysed turbulence models. Furthermore, that the prediction of forces or lift coefficient is significantly higher in the ANSYS Fluent software can be determined.

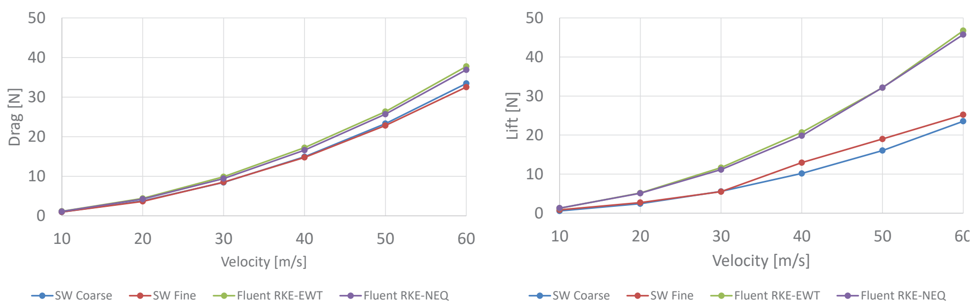

Figure 6: *Drag and Lift force versus velocity*

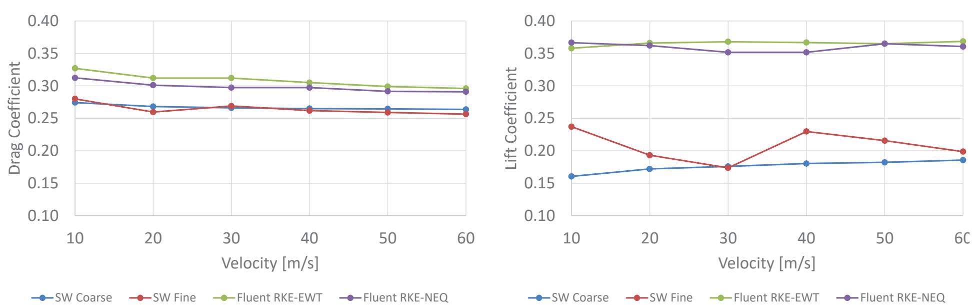

Figure 7: *Drag and Lift coefficient versus velocity*

Figure 8 shows a comparison between the drag coefficient and publicised measurements. A significant difference between the measurements and prediction with the SolidWorks Flow Simulation software in the complete analysis field can be determined. In contrast, both turbulence models used in the Fluent software give a very good prediction in the complete analysis area. Nevertheless, in the complete analysis area the same mesh was used, having 2.6 million final volumes. Table 2 displays a summary of the y+ variable on the walls of the body. In the analysis area, the value of the variable was between 1 and 163, and on average it was between 16 and 88.

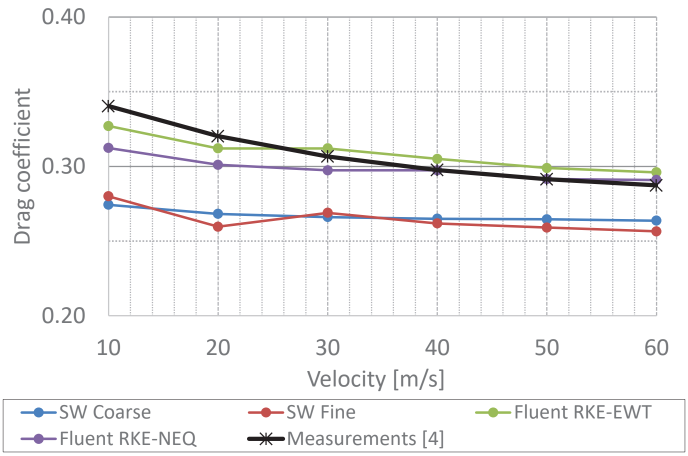

Figure 8: *Drag coefficient from simulations and measurements*

Table 2: *Variable y+ on the walls of the body at different velocities using the ANSYS Fluent software*

Velocity [m/s]	$y+_{min}$	$y+_{max}$	$y+_{ave}$
10	1	32	16
20	1	61	31
30	2	88	46
40	3.5	114	63
50	5	139	74
60	6	163	88

Figure 9 shows a comparison of the pressure field on the surface of the body and on the plane of symmetry. Despite the fact that at first glance the figures may appear very similar, from the analysis of lift and drag force, we know that there is a significant difference between the integrated value of the pressure distribution. Figure 10 shows a comparison of the velocity contours in both software programmes, namely on the plane of symmetry, as well as on the plane at an altitude of 0.244 m above ground level. In this case, a greater difference in prediction of velocity in front and behind the body can be observed.

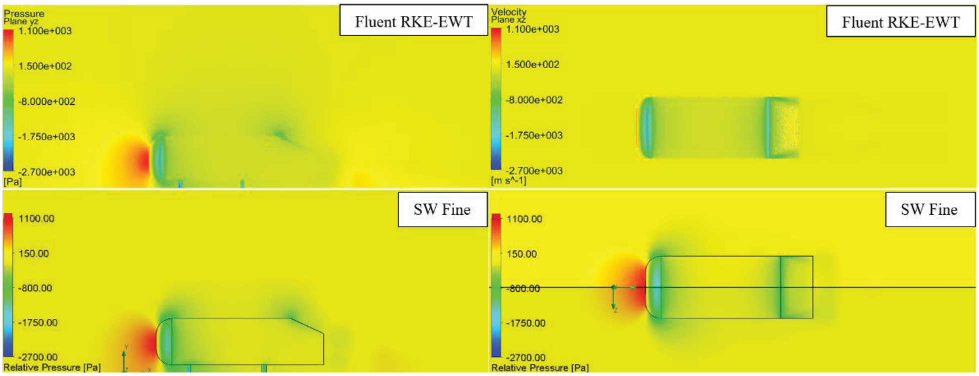

***Figure 9**: Prediction of pressure distribution at the velocity of 40 m/s*

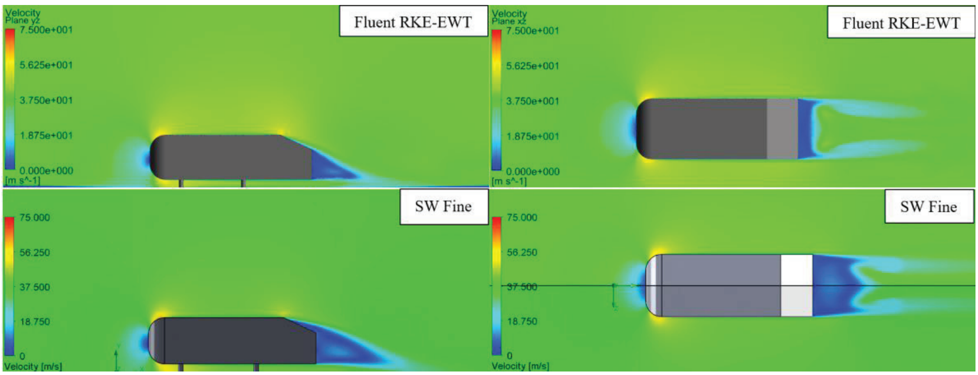

***Figure 10**: Prediction of velocity distribution at the velocity of 40 m/s*

Figure 11 shows a comparison of the turbulent kinetic energy on the plane of symmetry and on the plane at an altitude of 0.244 m above ground level. In this case, it is also obvious that there is a large difference in prediction between the analysed packages.

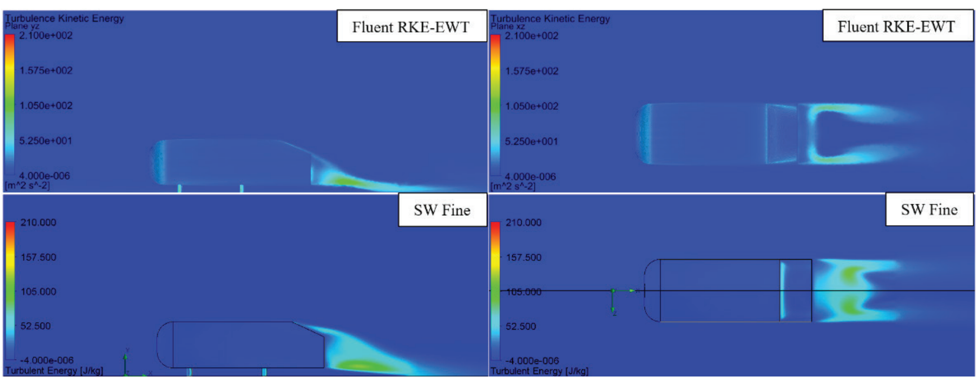

***Figure 11**: Prediction of turbulence kinetic energy at the velocity of 40 m/s*

Table 3 displays the computational times of individual analyses. For the analysis, a computer with an Intel i7 4790 CPU with 16 GB RAM was used. It is apparent from the table that the difference between computational times is significant; using the SolidWorks software, substantially more time for the convergence would be needed.

Table 3: *Computational times*

Computational time [hh]:[mm]				
Fluent RKE-EWT	Fluent RKE-NEQ	SolidWorks Coarse mesh	SolidWorks Fine mesh	SolidWorks adaptive mesh
0:40	0:33	2:40	17:10	65:00

6 CONCLUSIONS

The performed comparison analysis of the prediction of numerical analysis with the ANSYS Fluent software package, and the SolidWorks Flow Simulation software package shows a significant difference between the predictions of flow field. With the ANSYS Fluent software, a small deviation from the publicised measured values in a relatively short period is obtained. The SolidWorks Flow Simulation software package is significantly more inaccurate in prediction of results, and needs significantly more time.

The difference between the prediction among different packages can be explored in the technique of meshing as well as in the numerical algorithms themselves. The Cartesian mesh of the SolidWorks Flow Simulation software package did not enable sufficiently precise prediction of the forces on the Ahmed body, despite the extremely long computation time and the adaptive mesh technique. The trend of prediction in the analysed area otherwise follows the correct value, but the difference between the prediction and the measurement is large.

The performed analysis, both from the aspect of results accuracy as well as from that of computational times, shows that purpose-made tools for numerical analysis, such as ANSYS Fluent, are much faster and more accurate than add-ons to 3D modellers, as in the case of SolidWorks Flow Simulation.

References

[1] **I. Ščuri, M. Pezdevšek, I. Spaseski, M. Fike and G. Hren**, *"Numerical analysis of an axial flow fan: ANSYS vs SolidWorks,"* Journal of Energy Technology, vol. 4, pp. 11-19, 2014

[2] **I. Spaseski, M. Pezdevšek, I. Ščuri, G. Hren and M. Fike**, *"Numerical analysis of lift and pressure coefficient on fan airfoil: ANSYS vs SolidWorks,"* Journal of energy technology, vol. 3, pp. 63-73, 2014

[3] **S. Ahmed and G. Ramm**, *"Some salient features of the time-averaged ground vehicle wake,"* SAE Technical Paper 840300, 1984

[4] **W. Meile, G. Brenn, A. Reppenhagen, B. Lechner and A. Fuchs**, *"Experiments and numerical simulations on the earodynamics of the Ahmed body,"* CFD Letters, vol. 1, pp. 32-38, 2010

[5] **Y. Liu and A. Moser**, *"Numerical modeling of airflow over the Ahmed body,"* in Proceeding of the 11th Annual Conference of the CFD Society of Canada, 2003

[6] **S. Krajnović and G. Mineli**, *"LES investigation of the asymmetry in the wake of a generic vehicle body,"* in 4th International conference on Jets, Wakes and Separated Flows, ICJWS2013 IMECE 2013, 2013

[7] **C. Hinterberger, M. Garcia-Villalba and W. Rodi**, *"Large eddy simulation of flow around the Ahmed body,"* in The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains, Volume 1, 2004

[8] **M. K. A. B. Salleh**, *Simulation and analsys drag and lift coefficent between sedan and hatchback car.* Bachelor thesis, PAHANG: University Malaysia Pahang, 2009

[9] **M. Lanfrit**, *"Best practice guidelines for handling Automotive External Aerodynamics with FLUENT,"* 2005. [Online]. Available: http://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf. (25. 10. 2016)

Nomenclature

F_D	Drag force
F_L	Lift force
C_D	Drag coefficient
C_L	Lift coefficient
Re	Reynolds number
p_d	Dynamic pressure
A	Reference area
k	Turbulent kinetic energy
ε	Turbulent dissipation
ω	Specific dissipation rate
Δ	Relative difference
$y+$	Dimensionless wall distance
v	Air velocity