Journal of Industry, Engineering and Innovation, Vol. 1 No. 1 (2019) p. 1-10

JIEI

FAZ Publishing

Journal of Industry, Engineering and Innovation

Journal homepage : www.fazpublishing.com/jiei

e-ISSN : 2672-7382

Comparison between Solidworks and Ansys Flow Simulation on Aerodynamic Studies

Imadduddin Bin Ramlan[1], Nofrizalidris Bin Darlis[2]

[1 2]Department of Mechanical Engineering
Faculty of Engineering Technology
Universiti Tun Hussein Onn Malaysia
Pagoh, Muar, Johor

Email: imadduddinramlan@gmail.com

Received 00 Month 2000;
Accepted 01 Month 2000;
Available online 02 Month 2000

Abstract: Nowadays, with increase in competition in automobile sector, the Computational Fluid Dynamic (CFD) simulation have become one of important tool in the process of vehicle manufacturing. It is because the CFD simulation able to shorten the lead time in the research and development process. The idea of this project is to analyse the aerodynamic behaviour of a local sedan car, Perodua Bezza by using two different CFD simulation software which are SolidWorks Flow Simulation 2018 and ANSYS CFX 18.2. Both of the software used the same boundary condition and setup except for the type of flow used in the simulation in order to obtain a fair result. The velocity inlet used for this project is 30.56 m/s and the turbulence model selected is standard k-ε. The coefficient of drag for the model obtained for SolidWorks 2018 and ANSYS CFX 18.2 is 0.396 and 0.413 respectively. The percentage different of the drag coefficient between both software is about 4%, and this probably due to different type of flow used in this simulation. Compared to 0.286 drag coefficient value provided by the manufacturer, the percentage different between the SolidWorks 2018 and ANSYS CFX 18.2 with the manufacturer data is about 38% and 44% respectively. The different is might be due to simplified version of the car model, the parameters set is different with the manufacturer settings and also the type of flow used in the simulation. Overall, both software can be used for aerodynamic behaviours study but ANSYS CFX 18.2 is better choice because it provides multiple parameters settings which can be used to study more complex flow simulation.

Keywords: CFD, aerodynamic, SolidWorks, ANSYS CFX, coefficient of drag, Perodua Bezza.

1.0 Introduction

As the technology is developed, more software engineers have produced many simulation software in order to reduce the time taken for the research and development process and can save up more money. In addition, time is very crucial things in the most manufacturing company because they need to compete with others company to produce faster and quality product thus Computational Fluid Dynamic (CFD) software is very important in the automobile industry. There are two types of CFD simulation software which are open source type and commercial type. Most researchers are preferred to use a commercial type software such as SolidWorks and ANSYS to study the aerodynamic behaviours of a vehicle but the problem is which one is the best software to study the aerodynamic behaviours is still debateable.

The purpose of this technical paper is to compare the results of aerodynamic behaviour between two different CFD software which are SolidWorks 2018 and ANSYS. CFD is a branch of fluid mechanics that uses numerical methods with the help of computer to solve and analyse problems involving fluid flows [1]. CFD simulation technology helps engineers to understand the physical phenomena taking place around the design and provides an environment to optimize the performance with respect to certain criteria [2]. In automobile manufactures, CFD helps study the aerodynamics behaviour without having to create a physical model and thus it helps to reduce research and development costs while simultaneously saving time [1]. The use of CFD to predict aerodynamic flow around vehicles has been on the incline over the last few years
due to the increase in computing power available, making it a viable tool for simulating aerodynamic effects [3].

**Corresponding author: author@organization.edu.co*

Aerodynamics is the dynamic associated with studying the motion of air especially when it interacts with moving objects. Aerodynamics is a subfield of fluid dynamics and gas dynamics, with much theory shared between them [4]. Aerodynamics study in the automotive industry is very important in order to improve the performance, handling, safety, and comfort. Traditionally, wind tunnel tests are one of the process in vehicles design. In order to do the process, the vehicles or car body model should be made thus it will cost much money and time to complete the tests.

2.0 Literature review

2.1 Automotive design

Automotive design is not only focusing on the physical appearance only, but in the aerodynamics of the vehicles too. the main concerns of automotive aerodynamics are reducing drag, reducing wind noise, minimizing noise emission and preventing undesired lift forces at high speeds [5].

2.2 Aerodynamics theory

There are several theories that really important in the aerodynamics theory such as the Bernoulli's theorem, the aerodynamic parameters, and the process of aerodynamics design. In this section, all the theory will be discussed.

2.2.1 Bernoulli's theorem

The improvement on the characteristic related through the drag force which is ruled by Bernoulli Equation. It is one of the best-known and widely-used equations in fluid mechanics.

$$p + \tfrac{1}{2} pv^2 = \text{constant} \tag{1}$$

From equation (1) shows the increasing of velocity will case the decrease in static pressure and vice versa. The Bernoulli's Equation from equation (1) gives the important result which is:

Static pressure + Dynamic Pressure = Stagnation Pressure

2.2.2 Aerodynamic parameters

The drag coefficient (C_D) for a vehicle body can define as [6]:

$$C_D = \frac{D}{\frac{1}{2}pv_\infty{}^2 A} \tag{1}$$

Where D is the drag and A is the frontal area
Since, the C_D was defined as shown in equation (1). Thus, the drag force can derive as:

$$D = \frac{1}{2}\, pv_\infty{}^2 C_D . A \tag{2}$$

Besides that, the drag coefficient, C_{df} can derive from friction drag, D_f, on a flat plate as:

$$C_{df} = \frac{D_f}{\frac{1}{2}pv^2 bl} \tag{3}$$

Where D_f is friction drag, b and l are width and length of flat plate.

The lift force can be determined if the distribution of dynamic pressure and shear force on the entire body are known. Therefore, the lift coefficient (C_L) can indicate as:

$$C_L = \frac{L}{\frac{1}{2}pv_\infty{}^2 A} \tag{4}$$

Where L is lift force and A is the frontal area.
Pressure and shear stress distribution is difficult to obtain along a surface for non-geometry body either experimentally or theoretically but these to value can be obtained by Computational Fluid Dynamics (CFD) [6].

2.2.3 The process of aerodynamics design

Automotive aerodynamics is studied using both computer modelling and wind tunnel testing. In traditional ways, it requires more than one wind tunnel in the process of designing the car as shown in the Figure 2.1. Currently, almost all car manufacturers using the computational fluid dynamic simulation in the process of designing the car and use the wind tunnel for experimental purpose. In addition, the numerical simulation could provide enough data to the aerodynamic shape design of car [7].

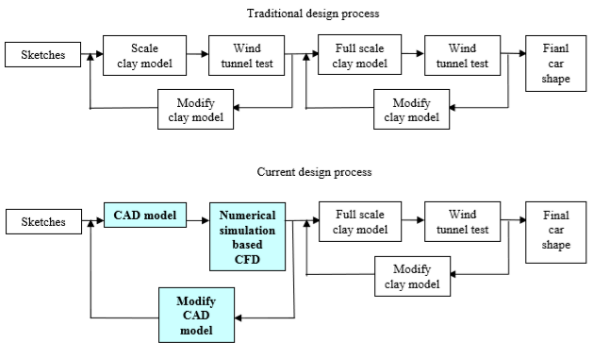


Figure 2.1: The process of car design [Source: 7].

2.3 Computational fluid dynamics (CFD)

Computational fluid dynamics (CFD) is concerned with numerical solution of differential equations governing transport of mass, momentum, and energy in moving fluids [8]. Today, CFD has become a very vital tool to predict flow movement and these techniques are found in almost all fields ranging from engineering to medical research [9].

2.3.1 Process flow

The basic procedure of CFD can be divided into three which are pre-processing, processing, and a postprocessor. The geometry modelling, grid generation, define model, set properties, set boundary and inlet conditions is a process in pre-processing. Then, simulation can be run and the equations are solved iteratively as a transient or steady-state. Finally, a post-processing is used to analyse the results and visualize the results. Steps in CFD simulations is illustrated in Figure 2.2.

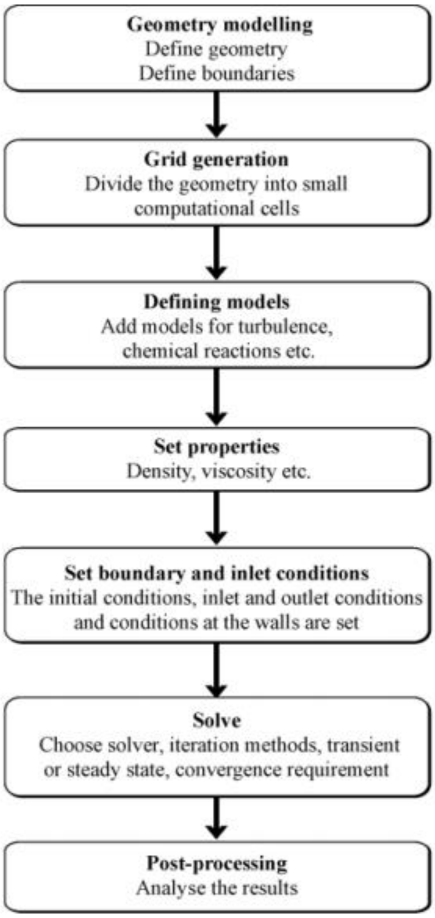


Figure 2.2: Steps in CFD simulations [10].

2.3.2 Advantages of CFD

Table 2.1: Advantages of CFD [11].

NO	TOPIC	ADVANTAGES
1	Accuracy	With increasing of high-speed computers, the resolution and cell size of CFD models has improved dramatically over the past few decades. Airflow Sciences Corporation, which has used both modelling methods since 1975, has made numerous comparisons between CFD modelling, physical modelling, and field testing. Results indicate that both types of models share the same accuracy when it comes to velocities and pressures.
2	Schedule	CFD modelling is almost always faster than physical modelling. In many cases, design results from a CFD model are available several weeks before similar results from a scale model.
3	Cost	CFD model studies are generally 20-40% less than a comparable physical model effort. This is tied quite strongly to the labour difference in model construction that influences the schedule. Also, many CFD tasks can be automated with the computer, including the design optimization process, whereas these tasks are primarily manual with the physical model.
4	Scale	Most physical models are built to scale, typically 1:12 or 1:16 for power plant models. CFD models are almost always built full size (1:1 scale). Care must be taken in computer models to ensure that the correct number, size, and shape of computational cells are used, and the level of detail to include must be considered in a scaled model to ensure geometric and dynamic similarity is maintained. In a CFD model, the Reynolds Number is often matched exactly, while in a physical model industry generally tries to match the Reynolds Number regime. Both are fine as long as the boundary layer is negligible. This is generally the case for large power plant duct systems. Note, however, that one must closely match the exact value of the Reynolds Number if the objective is to determine lift or drag characteristics, or any system where the boundary layer along a surface is important.
5	Storage	CFD models are usually stored on tape, CD-ROMS or DVDs which typically have a much longer storage life and negligible space requirements. Physical models can take up considerable space in a warehouse.

2.3.3 Disadvantages of CFD

Table 2.2: Disadvantages of CFD [12].

NO	TOPIC	DISADVANTAGES
1	Physical model	- CFD relies upon physical models of real-world processes. - CFD solution can only be as accurate as the physical models.
2	Numerical errors	- Solving equations on a machine also introduces numerical errors, example, round-off error (due to finite word size available on computer, round-off error will always exist) and truncation error (due to approximations in the numerical models)

3	Boundary conditions	- Similar to physical models - The accuracy of the CFD solution is only as good as the initial boundary conditions provided to the numerical model.

2.3.4 Software

Today, there are many commercial CFD programs available that use numerical analysis and algorithms to solve and analyse problems that involve fluid dynamics such as SolidWorks Flow Simulation, ANSYS Fluent, ANSYS CFX, Star-CD, FLOW-3D, COMSOL, Acusolve CFD, ADINA and ABAQUS [9] [10]. In this project, SolidWorks 2018 Flow Simulation and ANSYS 18.2 CFX will be focused on to study the aerodynamics of the car model.

2.3.5 Boundary condition.

The previous studies on CFD simulation using commercial software regarding the aerodynamic simulations is tabulated in Table 2.3.

Table 2.3: Boundary condition in previous study on CFD simulations.

Author/Year	Parab et al., 2014 [1]	Ramya et al., 2015 [13]	Dias et al., 2016 [2]	Taherkhani et al., 2017 [14]	Chaurasiya et al., 2017 [15]	Azmi et al., 2017 [16]
Software	ANSYS Fluent	ANSYS CFX	ANSYS Fluent	ANSYS Fluent	ANSYS Fluent	ANSYS Fluent
Inlet velocity	41.67 m/s	16.67 m/s	40.00 m/s	26.80 m/s	22.22 m/s	55.55 m/s
Pressure outlet	0 Pa	-	0 Pa	0 Pa	0 Pa	0 Pa
Type of flow	Steady-state	Steady-state	Transient state	Steady-state	Transient state	-

2.3.6 SolidWorks simulation vs ANSYS simulation

Table 2.4 shows the comparison between SolidWorks 2018 flow simulation and ANSYS CFX simulation in term of numerical method, turbulent models and the advantages and limitation of the software.

Table 2.4: Comparison of SolidWorks and ANSYS simulation.

Characteristic	**SolidWorks simulation**	**ANSYS simulation**
Numerical method	SolidWorks flow simulation solves the governing equations with a discrete numerical technique based on the finite volume (FV) method. Cartesian rectangular coordinate system is used. To obtain space discretization, the axis-oriented rectangular grid is used far from a geometry boundary. Thus, the control volumes as example mesh cells, are rectangular parallelepipeds. Near the geometry boundary Cartesian cut cells approach is used. According to this approach, the near-boundary mesh is obtained from the original background Cartesian mesh by cutting original parallelepipeds cells that intersect the geometry. Consequently, the near-boundary cells are polyhedrons with both axis-oriented and arbitrary oriented plane faces in this case. Thus, SolidWorks flow simulation combines advantages of approach based on regular grids and ones with highly accurate representation of geometry boundaries [17].	CFX is cell-vertex finite volume, coupled implicit, pressure-based solution technique (i.e., solves for pressure and velocity at the same time in the same A matrix). Pressure 8 and velocity are co-located, so p-v decoupling is dealt with using a Rhie-Chow approach. In vertex-based schemes the flow variables are stored at the vertices of the mesh elements. CFX uses an unstructured Finite Element based Finite Volume method. The FE basis comes from the use of shape functions, common in FE techniques, to describe the way a variable change across each element. It is also a node-based code, where the solution variables are solved and stored at the centers of the finite volumes, or the vertices of the mesh. [18].
Turbulent model	The modified k-ε turbulence model with damping functions proposed by Lam and Bremhorst (1981) describes laminar, turbulent, and transitional flows of homogeneous fluids consisting of turbulence conservation laws [19].	A number of models have been developed that can be used to approximate turbulence based on the Reynolds Averaged Navier-Stokes (RANS) equations. Some have very specific applications, while others can be applied to a wider class of flows with a reasonable degree of confidence. The models can be classified as either eddy-viscosity or Reynolds stress models. The following

		turbulence models based on the RANS equations are available in ANSYS CFX. Eddy viscosity models: Zero equation model, standard k-ε model, RNG k-ε model, standard k-ω model, realizable k-ε, SST zonal k-ω based model, Curvature correction for two-equation models. Reynolds stress models (RSM): Launder, Reece and Rodi Isotropization of Production model (LRR Reynolds Stress), Launder, Reece and Rodi Quasi-Isotropic model (QI Reynolds Stress), Speziale, Sarkar and Gatski (SSG Reynolds Stress), SMC- ω model (Omega Reynolds Stress), Baseline (BSL) Reynolds stress model, Explicit Algebraic Reynolds stress model (EARSM). CFX also provides the Large Eddy Simulation (LES) and Detached Eddy Simulation (DES) turbulence models [20].
Advantages	1. Easy modelling software for making parts and assembly. 2. User friendly. 3. Rapid results after analysis. 4. Integrated to different other software's like SOLID CAM, ANSYS. (Direct import and export of the file can be done easily).	1. Great software for analysis, allowing high node density 2. It has different turbulent models to choose for the simulation [21] [22].
	5. Flow simulation is integrated inside SolidWorks [21].	
Limitations	1. Not the best software for analysis. 2. The results not quite accurate [21].	1. Modelling is not that great. 2. Results take time to generate. 3. Not user friendly [21].

3.0 Research methodology

3.1 Flow chart

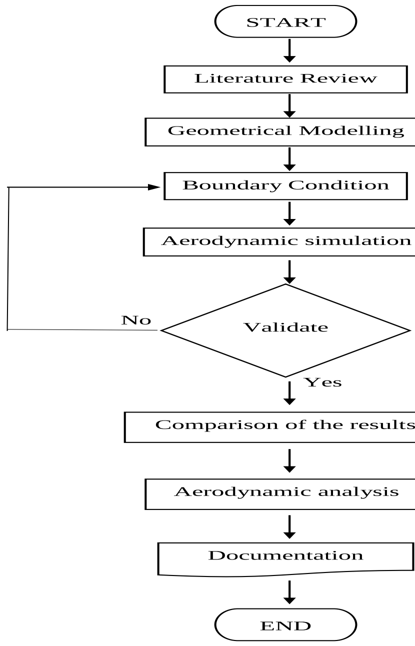


Figure 3.1: Flow chart.

3.2 Geometrical modelling

All the geometry of the car model was generated by using SolidWorks 2018 then the numerical solution was run by SolidWorks 2018 and also ANSYS CFX. The details of Perodua Bezza geometry are as follow:

Table 3.1: Perodua Bezza dimension. [Source: 23]

Overall length/ width/ height	4150/ 1620/ 1510 (mm)
Wheelbase	2455 (mm)

Figure 3.2 shows the shape of the car from the front, side and rear view. From the figure below, the sketch from picture method have been applied in order to design the car in SolidWorks followed by dimension stated in the Table 3.1.

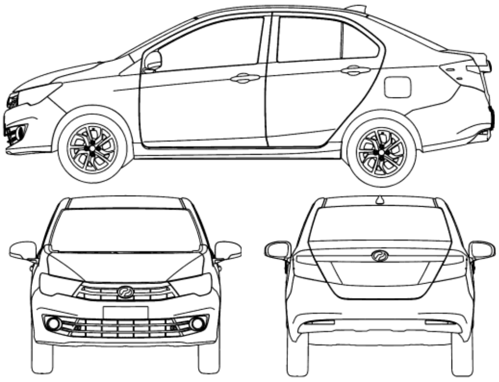


Figure 3.2: Perodua Bezza side, front and rear view.

Figure 3.3 show the drawing of Perodua Bezza after have been design in SolidWorks 2018.

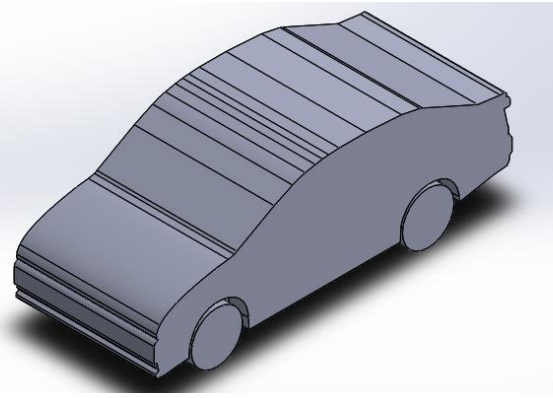


Figure 3.3: Geometrical model of Perodua Bezza.

3.3 Creating a fluid enclosure

In order to stimulate the air flow around the vehicle, a fluid volume needs to be created. This was done by creating a fluid enclosure around the model and subtracting the model. This fluid enclosure will act as the wind tunnel for the simulation. This fluid enclosure is created only for ANSYS CFX simulation, the fluid enclosure is created in the SolidWorks 2018 as shown in Figure 3.4 below. For SolidWorks simulation, the fluid enclosure does not need to be create because the analysis for SolidWorks is external flow, thus it only needs to adjust the computational domain size as same as fluid enclosure for ANSYS CFX simulation. The size of the enclosure L x W x H, Length equal to 7 car length, Width equal to 1 car length and Height equal to 2 car length.

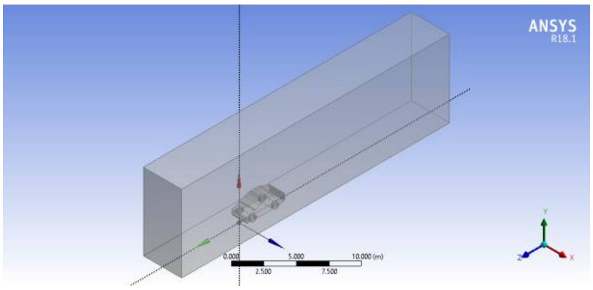


Figure 3.4: Fluid enclosure.

3.4 Meshing

For SolidWorks 2018, the mesh is generated by using automated mesh with highest level of initial mesh which is 7. For ANSYS CFX 18.2, the mesh is also generated automatically by choosing curvature-based mesh with angle of 10°. The figure below shows the mesh generated for SolidWorks and ANSYS CFX.

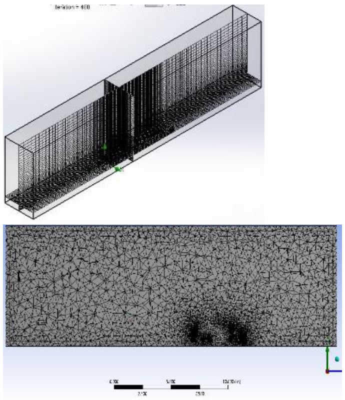


Figure 3.5: Mesh generated for SolidWorks (left) and ANSYS CFX (right).

3.5 Boundary condition

All the details of boundary condition are shown in the table below.

Table 3.2: Boundary condition.

Boundary types	**Value**
Inlet	30.56 m/s
Outlet	0 Pa
Wall	No slip wall
Fluid	Air at 25°C
Analysis type	Steady state solver
Turbulence model	k-epsilon model

4.0 Results and Discussions

The results presented in this technical paper is in table form or in figure form in order to make the readers to understand easily. In this section, the author will show the results and discuss about the comparison between SolidWorks and ANSYS in term of simulation, the validation for the simulation, the pre-processing, the solver and the post processing.

4.1 Validation of the simulation

The results for both CFD software was validated by comparing both CFD software results with the manufacturer data, as example the value of the coefficient of drag for both software was compared with the Perodua Bezza coefficient of drag which is 0.286. The results also validated by comparing both software results such as drag force, and the coefficient of drag and lift, the difference between both software results must be small in order to assume that simulation for both software is running with the same setup in order to make sure the results is valid to be compared.

4.2 Comparison between SolidWorks and ANSYS

4.2.1 Pre-processing

In this pre-processing, the meshing is one of the significant different for both software. The meshing details for both software are shown in the Table 4.2.1 below. The number of elements for both software is different because the SolidWorks does not have a wall to act as a wind tunnel, it only uses computational domain compare to ANSYS which use solid box with a specific dimension to act as wind tunnel and also a computational domain for the simulation, thus the same accurate and exact size for the computational cannot be done. Besides, ANSYS 18.2 managed to produce high number of elements and nodes because it can easily use the manual mesh
setup for the simulation compared to the SolidWorks manual mesh setup, thus the author used the automatic setup with the highest level of the initial mesh to match with the ANSYS mesh setup. Based on previous study, the highest the number of element and nodes, the accurate the results for the simulation.

Table 4.1: Details of meshing for both software.

Software	Number of elements	Number of nodes
SolidWorks 2018	512164	-
ANSYS CFX 18.2	662833	123916

4.2.2 The solver

Solver is where solution for the simulation takes place. The time needed for both software to complete the simulation is different. The iterations for the SolidWorks as the analysis run converged is 162 iterations, the iterations for the ANSYS as the analysis run converged is only 102 iterations. The higher the numbers of iterations, the longer the time needed for the analysis to be done. SolidWorks takes a lot of time to complete the analysis compared to ANSYS. The results for the post-processing will be discuss in the next sub-topic.

4.2.3 Post-processing

In this section, there are 3 results that will be compared and discussed which are the results of the coefficient of drag and lift for both software, the results of the pressure contour for both software and lastly the results of the velocity streamline for both software.

4.2.3.1 The coefficient of drag and lift

There are several things that need to be determine in order to calculate the coefficient of drag and lift such as the drag force that act on the car body on the front direction of the car and upper direction of the car, the frontal area of the car, the density of the fluid, and the velocity. Table 4.2 shows the value that is vital in calculate the coefficient of drag and lift. After determine all the value, just calculate the coefficient of drag and lift by using the equation stated in the literature review section. For SolidWorks, the user can set the equation goal to automatically calculate the coefficient of drag in which ANSYS CFX does not have the ability to do that. The results of the coefficient of drag and lift between two software is different. The coefficient of drag is really depending on the design of the shape of the model. The results of the coefficient of drag and lift for both software is shown in the Table 4.3 below.

Table 4.2: The value to calculate the coefficient of drag and lift of the model.

	SolidWorks 2018	ANSYS CFX 18.2
Drag force (z direction)	497.89 N	519.17 N
Drag force (y direction)	565.32 N	243.96 N
Frontal area of model (m^2)	2.196	2.196
The density of fluid (kg/m^3)	1.225	1.225
The inlet velocity (m/s)	30.56	30.56

Table 4.3: Coefficient of drag and lift for SolidWorks and ANSYS.

Software	Coefficient of drag	Coefficient of lift
SolidWorks 2018	0.396	0.450
ANSYS CFX 18.2	0.413	0.194
Percentage difference	4.29%	131.95%

The percentage different for the value of the coefficient of drag between SolidWorks and ANSYS with the manufacturer data are 38.46% and 44.41% respectively. SolidWorks has lower percentage different compared to ANSYS CFX. The reasons why the value for both software is far from the manufacturer data is due to simplified geometrical modelling of the car and also the different parameters used for the simulation compared to the manufacturer settings. The shape of the model is too simple due to lack of dimension details on that particular model and also to reduce computational time for the simulation thus it will affect the value of the drag force acting on the body of the car for all direction and also the value of the frontal area of the car. The massive different between both software for lift coefficient is because the different type of flow used for the simulation, SolidWorks 2018 used external flow to study the aerodynamic behaviour of Perodua Bezza and ANSYS CFX 18.2 used internal flow to study the aerodynamic behaviour of Perodua Bezza.

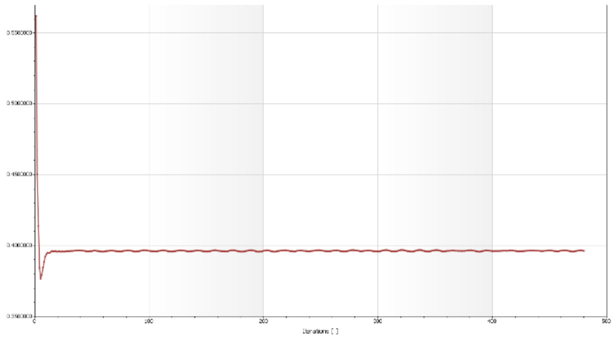


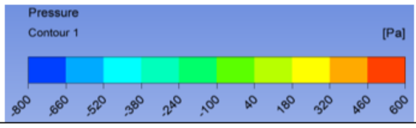
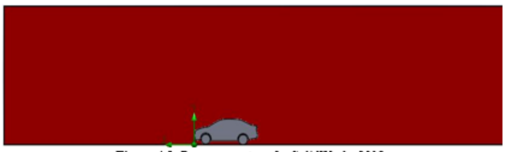
Figure 4.1: Coefficient of drag vs number of iterations for SolidWorks.

Another different between SolidWorks simulation and ANSYS CFX simulation is the ability to show the graph for coefficient of drag. SolidWorks can easily set the equation goal (coefficient of drag equation) and shows the chart for the equation goal but for ANSYS CFX, the user need to manually calculate the value of the coefficient of drag and manually produces the chart of coefficient of drag if needed. Figure 4.1 show the graph of coefficient of drag vs number of iterations for SolidWorks.

4.2.3.2 The pressure contours

Results of static pressure contours for both software will be shown in the figures below. Based on the results of simulation, it shows that both software has same higher-pressure concentration on the frontal area of the car. The pressure will drop when the air flows over the hood of the car and then increasing when the air flows reach the windshield. Then the pressure drops as the air flows travels over the windshield. The high pressure recorded for ANSYS CFX 18.2 is 0.58 kPa and 101. 94 kPa for SolidWorks 2018. The minimum pressure recorded for ANSYS CFX 18.2 is -1.34 kPa and 100.61 kPa for SolidWorks 2018. Based on Figure, the value for both software is very far however the location of the highest pressure at the body of the car is still the same. Based on previous studies, mostly the value of pressure (Pa) is in range between -3000 Pa to 2000 Pa, thus it can be concluded that ANSYS CFX 18.2 produce a better result in term of pressure value because the value is in range of the previous studies compared to SolidWorks 2018 which tend to produce high value of pressure.

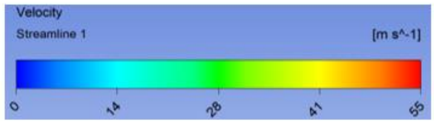
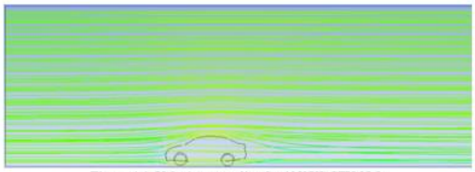
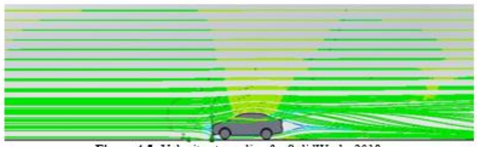
Table 4.4: Comparison of pressure contour between ANSYS CFX and SolidWorks.

Legend	
ANSYS CFX 18.2	 **Figure 4.2:** Pressure contour for ANSYS CFX 18.2.
SolidWorks 2018	 **Figure 4.3:** Pressure contour for SolidWorks 2018.

4.2.3.3 Velocity streamline

Figure below shows the velocity streamline for both SolidWorks and ANSYS CFX. For both velocity streamline, it shows that the air velocity decreasing as it approaches the front section of the car and the air velocity is higher at the hood section of the car. The highest air velocity recorded for ANSYS CFX 18.2 is 50.14 m/s and 37.57 m/s for SolidWorks 2018. The minimum velocity recorded for ANSYS CFX 18.2 is 0.07 m/s and 0 m/s for SolidWorks 2018. Both software has different results for its highest value of air velocity but the location of the highest air velocity is still the same. Both results is in range of the previous study, thus it shows that both software is relevant for this type of study.

Table 4.5: Comparison of velocity streamline between ANSYS CFX and SolidWorks.

Legend	
ANSYS CFX 18.2	 **Figure 4.4:** Velocity streamline for ANSYS CFX 18.2.
SolidWorks 2018	 **Figure 4.5:** Velocity streamline for SolidWorks 2018.

5.0 Conclusion

A Computational Fluid Dynamic (CFD) analysis was carried out by using two different software which is SolidWorks 2018 and ANSYS CFX 18.2 on a Perodua Bezza to analyse the aerodynamics behaviour for that car. The results of the simulation such as drag coefficient for both software were found to be very close, the percentage different between the result for drag coefficient is as low as 4.29%. However, the different between the value for both software and the manufacturer data is very high (38.46% and 44.41%). It is due the simplified version of the car model and also the parameters set by the author is nowhere near to the manufacturer settings due to unknown details. In addition, the geometrical modelling of the car needs to be improved in order to get more accurate results for the simulation. Based on the low percentage different between SolidWorks 2018 and ANSYS CFX 18.2 drag coefficient value, it can be concluded that SolidWorks 2018 and ANSYS CFX 18.2 is relevant in CFD simulation to study the aerodynamic behaviours of the car. Both software can provide good results if the details is set specifically. However, based on this study, ANSYS CFX 18.2 provide a better result to study the aerodynamic behaviours due to its various parameter settings and its flexibility compared to SolidWorks 2018. Both software have its own advantages and disadvantages such as SolidWorks 2018 provides a good user interface to design a model easily but the software is not very suitable to run flow simulation due to its limited parameters settings, on the other hand, ANSYS CFX 18.2 have a bad user interface to design a model but it can run a complex flow simulation due to its advanced parameters settings and flexibility.

References

[1] Parab, A., Sakarwala, A., Paste, B., Patil, V., & Mangrulkar, A. (2014). Aerodynamic Analysis of a Car Model using Fluent- Ansys 14.5. *International Journal on Recent Technologies in Mechanical and Electrical Engineering*, *1*(4), 7–13. Retrieved from http://www.ijrmee.org
[2] Dias, G., Tiwari, N. R., Varghese, J. J., & Koyeerath, G. (2016). Aerodynamic Analysis of a Car for Reducing Drag Force. *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE*, *13*(3), 114–118. https://doi.org/10.9790/1684-130301114118
[3] M Takagi. Application of computers to automobile aerodynamics. Journal of Wind Engineering and Industrial Aerodynamics, 33(1):419–428, 1990.
[4] Senger, S., Bhardwaj, S. D. R., & Bhard, R. (2014). Aerodynamic Design of F1 and Normal Cars and Their Effect on Performance, 4(4), 363–370.
[5] Kumar, K. (2016). A Review on Aerodynamic Study of Vehicle Body using CFD, (April 2014).
[6] Ahmad, (2008). Develop Drag Estimation on Hybrid Electric Vehicle (HEV) Model Using Computational Fluid Dynamics (CFD). Universiti Malaysia Pahang: Tesis Sarjana Muda.
[7] Zhang, Y., Zhang, Z., Luo, S., & Tian, J. (2009). Aerodynamic Numerical Simulation in the Process of Car Styling. *E-Engineering & Digital Enterprise Technology Vii*, *Pts 1 and 2*, *16–19*, 862–865. https://doi.org/10.4028/www.scientific.net/AMM.16-19.862
[8] Date (2005). *Introduction to Computational Fluid Dynamics*. Cambridge University Press, New York: Cambridge University Press.
[9] Darlis (2016). Improvement of Spiral Flow Aortic Cannula for Cardiopulmonary Bypass Operation. Universiti Teknologi Mara: Tesis Doktor Falsafah.
[10] Andersson, B., Andersson, R., Hakansson, L., Mortensen, M., & van Wachem, B. G. M. (2012). *Computational fluid dynamics for engineers*. *Austin, TX: Engineering Education System*, *1989*. Retrieved from http://www.csa.com/partners/viewrecord.php?requester=gs&collection=TRD&recid=A9542450AH
[11] Linfield, K. W., Robert, P. E., & Mudry, G. (2008). Pros and Cons of CFD and Physical Flow Modeling. *Airflow Sciences Corporation*. Retrieved from http://www.airflowsciences.com/sites/default/files/docs/Pros-and-Cons-of-CFD-and-Physical-Flow-Modeling.pdf
[12] What Is Computational Fluid Dynamics (CFD)? Application & Advantages. (2018, May 09). Retrieved from https://redmetal.co.za/engineering-services/computational-fluid-dynamics-flow-simulation/
[13] Ramya, P., Kumar, A. H., & Ramanaiah, J. M. N. (2015). Analysis of Flow over Passenger Cars using Computational Fluid Dynamics, *29*(4), 170–176.
[14] Taherkhani, A. R., Gilkeson PhD, C., Gaskell PhD, P., Hewson PhD, R., Toropov PhD, V., Rezaienia PhD, A., & Thompson, H. (2017). Aerodynamic CFD Based Optimization of Police Car Using Bezier Curves. *SAE International Journal of Materials and Manufacturing*, *10*(2), 2017-01-9450. https://doi.org/10.4271/2017-01-9450
[15] Chaurasiya, V. V, Kushwaha, D. B., & Raees, M. (2017). Aerodynamic Analysis of Vehicle Using CFD. *International Journal of Recent Trends in Engineering & Research*, 131–137. https://doi.org/10.23883
[16] Azmi, M. F. M., Marzuki, M. A. B., & Bakar, M. A. A. (2017). Vehicle aerodynamics analysis of a multi-purpose vehicle using CFD. ARPN Journal of Engineering and Applied Sciences, 12(7), 2345–2350.
[17] Lai, Y. (2009). Technical Reference. *Imid 2009*, (159679), 1069–1072. https://doi.org/10.1590/S0034-73292005000200004
[18] What is the difference between Ansys cfx and fluent? (n.d) Retrieved from https://www.quora.com/What-is-the-difference-between-Ansys-cfx-and-fluent
[19] Sobachkin, A., & Dumnov, G. (2014). Numerical Basis of CAD-Embedded CFD. *NAFEMS World Congress 2013*, (February), 1–20. https://doi.org/10.1007/s10590-014-9157-9
[20] Turbulence and Near-Wall Modeling (2015, June 29). Retrieved from https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/cfx_mod/i1345898.html
[21] What do you think is better: ANSYS or Solidworks? (n.d) Retrieved from https://www.quora.com/What-do-you-think-is-better-ANSYS-or-Solidworks
[22] Fluent Vs Solid-works Flow Simulation. (n.d.). Retrieved from https://www.cfd-online.com/Forums/main/89831-fluent-vs-solid-works-flow-simulation.html
[23] Bezza Specification. (n.d.). Retrieved from http://www.perodua.com.my/specification/bezza