APPLICATIONS OF OPEN-SOURCE CAE TOOLS IN MECHANICAL DESIGN

NOOR FARAHANA BINTI ABDUL RAHMAN

Final Report

Projek Sarjana Muda II

Supervisor: DR. SYAMSUL ANUAR BIN SHAMSUDIN

Second Examiner: EN. NAZIM BIN ABDUL RAHMAN

Faculty of Mechanical Engineering

Universiti Teknikal Malaysia Melaka

MAY 2017

APPLICATIONS OF OPEN-SOURCE CAE TOOLS IN MECHANICAL DESIGN

NOOR FARAHANA BINTI ABDUL RAHMAN

This report is submitted in fulfillment of the requirement for the degree of Bachelor of Mechanical Engineering (Design and Innovation)

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

MAY 2017

APPROVAL

“I hereby declare that i have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of mechanical Engineering (Design & Innovation)”

Signature :

Name : DR. SHAMSUL ANUAR BIN SHAMSUDIN

Date :

DECLARATION

I declare that this report entitle “Application of open-source CAE tools in mechanical design” is the result of my own research except as cited in the references.

Signature :

Name : NOOR FARAHANA BINTI ABDUL RAHMAN

Date :

DEDICATION

To my beloved my father, Abdul Rahman Bin Ghani, my mother, Siti Alina Binti Ahmad, my younger brother, Muhammad Hafizat Bin Abdul Rahman, supervisor and friends

ABSTRACT

The purpose of this project is to study the applications of open-source Computational-Aided Engineering (CAE) tools in mechanical design. The Computational-Aided Engineering type that has been chosen is Computational Fluid Dynamics (CFD); and the model design that fulfill the criteria to be simulated in this project is the part of aerodynamics model. This aerodynamics design is synonym in industrial designing because of the feedback in terms of its accessibility, convenience and safety. In the creation of model aerodynamic concept, there are a lot of things that need to be identified and calculated as it is hazardous and need to supervise from a professional. As this project research is to study on how work the simulation on certain chosen software that has been applicate on the selected model design. There are two type of software has been selected that are open-source software and commercial-package software. The preferred software is intentionally has been elected in order to see the different in the type of used, the meshing and airflow direction. The software that has been picked for open-source software are OpenFOAM and Gmsh, while for commercial-package are ANSYS-Fluent and SolidWorks. This all software is being using to analyze the CFD simulations and understand the procedure in using it.

ABSTRAK

Tujuan bagi menjalankan projek ini adalah untuk mengkaji aplikasi bagi sumber terbuka (*open-source*) dengan menggunakan alat pengiraan kejuruteraan dibantu (*Computational Aided Engieering*) di dalam reka bentuk mekanikal. Jenis pengiraan kejuruteraan dibantu (*Computational Aided Engineering*) yang telah di pilih ialah pengiraan dinamik bendalir (*Computational Fluid Dynamics*); dan reka bentuk model yang memenuhi kriteria untuk simulasi dalam projek ini adalah sebahagian daripada *aerodinamik* model. Reka bentuk *aerodinamik* ini adalah sinonim dalam prindustrian pemprosesan reka bentuk berdasarkan maklum balas dari segi akses, kemudahan dan keselamatan. Dalam penciptaan konsep model aerodinamik, terdapat banyak perkara yang perlu dikenal pasti dan dikira kerana ia adalah berbahaya dan perlu penyelia daripada pakar. Penyelidikan projek ini ialah untuk mengkaji betapa proaktifnya simulasi pada perisian yang dipilih untuk di gunakan sebagai bahan eksperimen kepada reka bentuk model yang dipilih. Terdapat dua jenis perisian yang telah dipilih iaitu perisian sumber terbuka (*open-source*) dan juga perisian sumber berbayar (*commercial-package*). Perisian yang dipilih adalah dengan syarat yang telah dipenuhi di dalam syarat pemilihan adalah untuk melihat perbezaan dalam bentuk cara penggunaan, gambaran bersirat dan arah aliran udara yang beraksi di reka bentuk model yang dipilih. Perisian yang dipilih untuk sumber terbuka (*open-source*) ialah OpenFOAM dan Gmsh, manakal untuk berbayar (*commercial-package*) ialah ANSYS-Fluent dan SolidWorks. Analisis pengiraan dinamik bendalir (*Computational Fluid Dynamics*) dan cara penggunaan telah dianalisis menggunakan keempat-empat perisian tersebut.

ACKNOWLEDGEMENT

At first, “Alhamdulillah” and without Allah’s will my *Projek Sarjana Muda* (PSM) report would not be completed. Then, I would like to express my special thanks to my parents Mr. Abdul Rahman Bin Ghani and Mrs. Siti Alina Binti Ahmad for their assistance without limits and all the motivations that they have gave throughout my difficulties along completing this project and report. Next, I would like to thank to my supervisor, Dr. Shamsul Anuar Bin Shamsudin that always supporting and guiding me in guidance, stimulating suggestions and constant encouragement that help me to finish my project especially in writing report. I also appreciate the guidance given by En Nazim Bin Abdul Rahman especially during this project presentation of VIVA and report that improved my presentation skills with thier comment and tips. Moreover, I would like to thank my laboratory, who gave me the permission to use all the machienery and all the facilities in order to finish my project report. Last but not least, I wish to avail myself of this opportunity to express a sense of gratitude and love to my family and friends for their help, moral support, and encouragement in everything.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

ANSYS	Analysis of System
CAD	Computer-Aided Design
CAE	Computer-Aided Engineering
CFD	Computational Fluids Dynamics
CLI	Command Line Interface
FEA	Finite Element Analysis
Hex	hexahedra
IGS/IGES	Initial Graphics Exchange Specifications
NASDAQ	National Association of Securities Dealers Automated Quotations
OBJ	Object
OpenFOAM	open source for Field Operation and Manipulation
Split-hex	split-hexahedra
STL	Stereo lithography
STP	Standard for the Exchange Product

LIST OF SYMBOL

m/s	Meter per second
kPa	Kilo Pascal
K	Kelvin

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

This chapter will label about the idea in this investigation. This idea was introducing by targeting to solve the problem in how to decrease the cost spending in engineering software. In this section also will label about the background of this study, the problem statement encouraged to this enquiry, objective and scope take account in this study.

1.2 BACKGROUND

This research is focusing more in Computational Fluid Dynamics (CFD) analysis. Nowadays, engineering field especially in design department does pay attention and rely a lot on simulation analysis before certain products were produce. Simulation analysis is playing an important role in production costing because it will lessen the cost in prototype production. Besides, using simulation analysis is easier in checking the thermal, pressure, and fatigue analysis by using just one software and the design can be optimizing after the fatigue part were analyze.

In order to making this research, little simulation analysis software's function is being identified and recognize. There are two type of license on software that is open source and commercial packages. Commercial packages are a software that provide pay license while, the other software that is open source does not need paid license because the license can be downloaded directly from the internet. Of course, the paid one will give a lot of function in simulation because it is marketing, everything does not come free and every upgraded version of software will come with expensive prices.

In this research, the simulation analysis was focus just in Computational Fluid Dynamics (CFD). The CFD simulation analyses were focused on both open source and commercial packages license. Every permit will be studied about two examples of software; for open source software: ANSYS-Fluent and SOLIDWORK while for commercial packages: OpenFOAM and OpenFVM.

Most of the commercial package usually can be bought or downloaded directly from internet but needed to pay; it is also coming in the compiled ready-to-run version. The meaning of compiled is the actual program code that the developer created, known as the source code, has run through a special program called a compiler that translates the source code into a form that the computer can understand. It is difficult to modify the compiled version of most applications and nearly impossible to see exactly how the developer created the parts of the coding inside the program. (Initiative, 2013)

The source code is included with the compiled version, so the user can modify and edit the coding to be more useful for long time used. The software developer who developed the open source concept with license that not interfere with the operation of other software are allowing the user to modify the source code are causing the application will be more useful and error-free over the long term. For example, as Red Hat has done with Linux. (Lakhani & Von Hippel, 2003)

1.3 PROBLEM STATEMENT

Currently, the cost of engineering software like ANSYS-Fluent, Computational Aided Design (CAD), and SOLID-WORK have risen (Menter, 2012). In order to low the cost of production, the simulation analysis be present on the rare design. The problem is in a way to complete the analysis, the suitable software needed. Towards fulfill the analysis requirement by technology; the software was come with new version. The difficult quantified is new version comes with luxurious price.

1.4 OBJECTIVES

The objectives of this research are to:

1. Differentiate between open source and a commercial package.
2. Compare the Computational Fluid Dynamics (CFD) analysis of open source and a commercial package.

1.5 SCOPE

The scope of this study is primarily focuses on Computational Aided Engineering (CAE) especially on Computational Fluid Dynamics (CFD) analysis. Principle open source and commercial package software was explored in this research. Besides, the procedures and different between commercial package software and open source software are studied. The simulation is focusing in Computational Fluid Dynamics (CFD) analysis and the other simulation is not included in this research. The coding of open source software is not being studied and does not been changed along this research. This research just focusses on testing the software and get the illustration of the Computational Fluid Dynamics (CFD) simulation.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

In this section, the parts of the car that have been undergoes testimony in flow direction are chosen. Further, the open source and commercial packages software capabilities are studied in order to analyze the capabilities in identified the flow direction on the product with approximate reading. Finally, the description of the software is presented.

2.2 COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

Computational Fluid Dynamics (CFD) is one of the fluid mechanics that uses of both numerical analysis and algorithms as converter to real flows simulation to be analyzed. In engineering field especially in aircraft and structure engineering, Computational Fluid Dynamics analysis is widely in used. Based on (Dynamics et al., n.d.), the experiment methods is important in confirming the validity and limits of approximations to the governing equations. Basically, the flow involving governing equations are extremely complicated, for example analytic solutions is possible to be solve for most practical applications. CFD introducing computational techniques as solutions, which replace partial differential equations with systems of algebraic equations which is much easier to solve using computer. In late 1950's, the improvement in computing power solutions become steady and its use is becoming increasingly rampant. (Collins, n.d.)

Next, CFD provides both qualitative and sometimes even quantitative prediction of fluid flows which means of mathematical modeling (partial differential equations, numerical methods (discretization and solution techniques), and software tools (solver, pre- and post-processing utilities). CFD also enables engineering and scientists to perform virtual flow laboratory in computational simulations. (Collins, n.d.)

Figure 2-1 Real experiment virtual

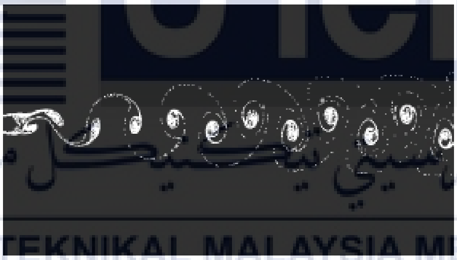


Figure 2-2 CFD simulation virtual

Most of CFD problems are solved using the Navier-Stroke equations that is define single phase which is gas or liquid as fluid flows. These equations can be simplified and lead as new equations as the term in the main equations being removed. When the term describing viscous actions to yield is removed from the main equations, the new equation is defined as Euler equations. Next, by simplify the equations with removing the term related to vortices yield make the equations be the full potential equations. (Kuzmin, n.d.)

2.2.1 PHASE IN COMPUTATIONAL FLUIDS DYNAMICS (CFD) ANALYSIS

In engineering fields, engineers commonly predict the behavior of the systems to facilitate understanding the relationship between the system variables. This method is important to design a better system for optimizing the operations. Usually, engineers perform the experiments directly or construct certain mathematical models that represent the system which contagion concerning to understand the system directly.

Next, another steps to include during the prediction method is constructed a mathematical model based on the understanding of the basic physical phenomena of the system. Understanding of system behavior can assist engineers in given certain conditions to find a mathematical solution by resulting system of equations. This step can be labeled as analytical approach.(Lomax, Pulliam, & Zingg, 1999)

Another approach can be describing is use CFD methods. A CFD method is described as the differential equations that govern the system is replaced with a set of algebraic equations at discrete points. The equations then are solved using digital computers. (Dynamics et al., n.d.)

2.2.1.1 EXPERIMENT TECHNIQUES

Most of reliable experiment data and information are getting from the measurement. In certain experiment involving full-scale equipment can give clue about hoe the equipment can perform under certain conditions. However, most of practical engineering applications does not involve full-scale equipment; full-scale test basically is either very expensive or difficult to handle or not possible at all. Full scale equipment is defined as the scale or size of the testing product are design or create exactly the same as an original product. So, an alternative solution is to perform the experiment on the small scale models. As can relate with this paper is, the original of the model is big and to create with real material and design as the original also high in cost production, so the solution is produce car body frame for same design but in small scale.

In this all advantages stated before, there is also disadvantages in using this experiment method as identify the flow directions manually. Although the model that has to be tested can be created in small size in order to save the cost; the resulting information of the small scale models needs to be extrapolated to the original models. The small scale models can also be models to be tested, but analysis of the testing is limited. Small scale models usually not simulate all the features of the original scale models; this will limit the usefulness of the test result. All of this situations, the most difficult stage in experimental method is when there are significant errors in measuring equipment. It will affect the analysis result and it will cause the major changing of the original design. From this disadvantage, it does not mean that computational models is paramount importance; it should be stress that the validation of the experimental data of numerical models is importance before it is put to good use.(Center, 2015)

2.2.1.2 ANALYTICAL METHODS

An analytical model is the consequences of mathematical models that represent the characteristics of certain system. In certain systems, it is too complex to be translated by drawing; so the function of mathematical model is translating the complex system into model that representing the physical process mainly consists of a set of differential equations. In most practical engineering applications, a lot of assumptions and simplifications are made to enable the analytical solution of differential equations created representing physical situations.

The assumptions and simplifications made in analytical solutions of the differential equations will limit the applicability of these methods. When there are too many assumptions and simplifications in system will limits the applicability of these methods to simple type of problems, although it will limit the validity of the solutions. (Center, 2015)

Despite that, analytical models played significant role in making the engineer or scientist understanding the fundamental of the rules of the system. In addition, this method played role as first stage in the validations of CFD models.

2.2.1.3 COMPUTTAIONAL FLUID DYNAMICS (CFD) TECHNIQUES

CFD technique is one method that involves the advent of digital computers. Computer such high electronic devices, that can calculate or solve high level equations in short time. Using this method, a large number of numerical methods were developed to solve flow problems. The purpose of flow simulations is to analyze the behavior of the flow in a given system for a given set of inlet and outlet conditions. This condition is known as boundary conditions.(Lomax et al., 1999)

Using this method also can identify the flow pattern and temperature distribution within the system; from the analysis, the design improvement need to be made to improve the defect of the system. The fundamental of CFD methods is to fine the values of flow quantities at a large number of points spread around the system geometry. These points are usually connected together and perform as numerical grid or mesh. Based on (Center, 2015), CFD concepts is replaced the system of differential equations with system of algebraic equations; in another words is the flow is converted to interdependency of the flow at those points and another points surrounding.

The development of fast and validated numerical procures, and increasing in computer speed, larger problems are being solved using CFD methods at cheaper cost and in a short time. Based on past study (Lomax et al., 1999), CFD methods are quickly replacing the experimental and analytical methods in many design and analysis applications. In addition, compared to experimental procedures in most engineering applications, CFD methods offer more complete set of information's in speed and reduced cost. It is usually providing all relevant flow information throughout the domain of interest. (Center, 2015)

CFD simulations are enable engineer or scientist to set the data of the flow solutions at true and original scale with the actual operating conditions. Compared to experimental and analytical techniques, CFD methods provide realistic conditions with economically represented and the results can be obtained directly.

2.2.2 THE APPLICATIONS OF COMPUTATIONAL FLUIDS DYNAMICS (CFD) ANALYSIS

Nowadays, the most widespread CFD software such as ANSYS-FLUENT and ANSYS-CFX are used commonly in the industrial applications. In this paper, the CFD can be used to stimulate the flow over a vehicle. For instance, the flow tested can be used to study the interaction between the speed of the car and the car body. The followings figure shows the prediction of the pressure induced by the interaction between the speed of the car and the car body. The models of the car body can be represented with models of varying complexity.

The pressure contour and a cutaway view that present on the result of the tested will be the datum in actual design improvement in order or to embrace the safety requirement being the first factors to be considered. CFD is attractive to industry since it is most cost-effective compared to physical testing. However, the engineering expertise is needed to obtain the validate solutions when the flow simulations are too complex and error-prone. In a large number of areas including engine components, auxiliary systems and modeling the aerodynamics of the car to minimize drag and optimize down force under operating conditions.

2.2.3 STAGES IN COMPUTATIONAL FLUID DYNAMICS (CFD)

In CFD simulations, there are three main stages that are pre-processing, solving and post-processing. In pre-processing stages, the problem faces are studied and identify to get the enough information before any solutions were listed. As the first step, the formulation of the problem such governing equations and boundary conditions are decided. Related to this paper, the boundary conditions a set inlet and outlet of the flow is tested on the model. After boundary conditions being analyzed, construct a computational mesh by setting the constant value of control volumes.

Next, the second stage is about solving the problems by using the data information collected from pre-processing stages. From the data before, the governing equation was analyzed and the algebraic equation was resulting to get the solution. The resulting of algebraic equations then be analyzed and solve using CFD methods.

The last stage in CFD simulations is post-processing. Post-processing is where the result of the analysis is being studied. Calculations of derived quantities such as forces and flow rates is defined and shown. The visualization such as graph and plots of the solutions also being shown in a digital computer those provide CFD simulations. From the visualizations using CFD software such as ANSYS-FLUENT and OPENFOAM, the flow direction shown in many colors with different meaning. The highest velocity in flow directions will label with red colors and the lowest velocity flow will label with dark blue color. From this visualization, the part with highest and lowest in velocity of the flow direction can be identified.(Center, 2015)

2.3 SOLIDWORKS

SolidWorks is a solid modelling computer-aided-engineering design (CAD) and computer-aided-engineering (CAE) that have been programmed as the computer program that can be running on Microsoft Windows. SolidWorks is published by Dassault Systèmes.

In December 1993, the SolidWorks Corporation was founded by Massachusetts Institute of Technology graduate Jon Hirschtick. On the first stage, Hirschtick recruited a team of engineers programming with the goals of constructing 3D CAD software that was easy to use, reasonably priced, and can be runs on the Windows. The first product SolidWorks 95 was launched in November 1995.

SolidWorks was already market several versions of the SolidWorks CAD software in e-Drawings, a collaboration tool, and DraftSight a 2D CAD product. Historic stated that SolidWorks is the world's most popular CAD software. The user is including from individuals to large companies and covers a world-wide cross-section of manufacturing market segments. SolidWorks also have partners with third party developers to add functionality in the market requests like finite element analysis, circuit layout, and tolerance checking; otherwise, SolidWorks also have license on its 3D modeling capabilities to the other CAD software retailers, remarkably ANVIL.

2.3.1 SOLIDWORKS FEATURES

2.3.1.1 DRAWING AND ASSEMBLY

SolidWorks is a solid design modeler that operates a parametric feature-based method to generate models and assemblies. The software is written on Parasolid-kernel and the parameter chosen are refer to the limitations whose standards regulate the shape or geometry of the model or assembly. The parameters can be either numeric parameters: line lengths or circle diameter, or geometric parameter: tangent, parallel, concentric, horizontal or vertical. Numeric parameter is designing to be related with each other through the use of associations which allows them to catch design intent.

SolidWorks provide the shapes and operations that construct the part; shape-based features usually begin with 2D or 3D sketch of shapes like circle, holes and slots. This shape is then being extruded or cut to add or remove material from the part. Operations-based features are not sketch-based and including features such as fillets, chamfers, shells, applying draft to the faces of a part that adding part or remove part in order to complete the design finishing.

Assembly is a solid-design process that combine all the part drawing; the analog to sketch relations are mates. Assembly-features provide sketch relations that define conditions as tangency, parallelism, and concentricity to the sketch geometry. Assembly mates define as equal associations with respect to the separate parts or components by allowing the easy structure of assemblies. SolidWorks also give an additional advanced features that allow mating features such as gear and cam follower mates, which provide designated gear assemblies to accurately imitate the rotational movement of an actual gear train.

At the end, the drawing design can be created either from parts and assemblies in printed copy by convert the sketch to drawing module. Views is automatically produced from the solid model and notes, dimensions and tolerance can be easily pop up to the drawing as wanted. The drawing can be presented on most paper sizes and standards like ANSI, ISO, DIN, GOST, BSI and SAC.

2.3.1.2 FLOW SIMULATION

SolidWorks have presented about the 3D solutions in flow simulations by Computational Fluid Dynamic (CFD) that available as a separately purchased product that can be used with SolidWorks Standard, SolidWorks Professional, or SolidWorks Premium.

SolidWorks Premium provides an add-in to SolidWorks with Circuit Works that allow the user to create a 3D models from the file formats written by most electrical computer-aided design (ECAD) system. SolidWorks also collaborating to design printed circuit boards (PCBs) that fit and function in SolidWorks assemblies. The user of SolidWorks also has a permission to export multiple SolidWorks Flow Simulation plots in a single e-Drawing file.

2.3.2 SOLIDWORKS SIMULATION

2.3.2.1 POST-PROCESSING

Dassault Systemes has been added new feature in SolidWorks that is post-processing simulations; this features provide a common access to edit definition, chart option and setting property managers.

In this simulations, SolidWorks offers the mirror results about planes symmetry and results comparison across the configurations. The result comparison across configuration provide with the compare result tools that allow the user to compare up the results plot from the simulation studies. Simulation studies offer the users with the different configurations of the same models.

Next, the mirror result abut planes symmetry is describing about the models that apply planar symmetry or cyclic symmetry restraints to control the geometric and loading symmetry conditions; instead the full body can be presented on result view that give the better insight on the model behavior and the identity of modelling errors.

2.3.2.2 MESHING

Meshing is one of the Finite Element Analysis (FEA); its providing a reliable numerical technique for analyze the design of engineering. The process of meshing is started from the program subdivides the model into small pieces or simple shapes that name as element connected at common points called nodes. Meshing process is where the model are undergoes subdividing into small pieces and finite element analysis program will depends on the model as network of interconnected elements.

The quality of the mesh will affect the accuracy of the solution and the finer the mesh. The better the accuracy. The generated mesh will be depending on the following factors:

- The created geometry.
- The options of active mesh.
- The conditions of the contact
- The global size and mesh tolerance.

2.3.2.3 PERFORMANCE

The performance of SolidWorks has been improved and be better from the previous performance. In the upgraded performance of SolidWorks provides that reducing the time solutions and be better convergence of results for non-linear studies, initial loading of simulations studies and the availability of new direct sparse solver for large problems.

The upgrading of the performance in non-linear studies that has been included is:

1. The overall performance development is upgrading up to 30% in merging for node-to-surface no-penetration contact formulation.
2. The accuracy of result for shells with plasticity material models when using small displacement formulation is improved.

Next, the other performance that has been improved is initial loading of simulation studies. The time for performing initial loading of models with simulation studies is reduced and the multiple simulation studies performance is more noticeable.

2.4 ANSYS

In 1970, ANSYS was founded by Dr. John A. Swanson; the headquartered of ANSYS is in United States, south of Pittsburgh in Canonsburg, Pennsylvania. ANSYS Inc. is Computer-aided engineering software that played a critical role as the global leader in engineering simulation. Based on (Inc., 2006), the example of ANSYS software being a critical role in product creation are rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device and crossed a bridge.

ANSYS was published as commercial software license that consist of range of disciplines including finite element analysis, structural analysis, computational fluid dynamics, explicit and implicit methods and heat transfer. ANSYS played as main role in production of certain product by verifying new product for how it will works before ever making a prototype. It will save the cost and time while ease and perform flawlessly service in product development.(Inc., 2006)

ANSYS certainly become important in engineering field especially design and innovation course. Every product analysis will be analyzed by different software beneath ANSYS. Although ANSYS have a lot of superiority in design and analysis process, this research will focus only on fluids dynamics. Example software ANSYS for computational fluid dynamics is ANSYS-Fluent. This software tools can stimulate fluid flows in a virtual environment – for example, the fluid flow in vehicles aerodynamics.

2.4.1 ANSYS-FLUENT

ANSYS-Fluent software is software that needed predictable of data of the new product based on the broad physical modeling. Originally, the idea of ANSYS-Fluent production was created by Dr. Ferit Boysan, researcher pioneered technology at Sheffield University in United Kingdom, Bart Patel, head department of Creare Inc., and Michael Engelman, major player in finite element CFD; the ideas from developing Fluent is about easy-to-use and interactive CFD software code for engineers. This collaboration between Sheffield University and Creare Inc. advocate the launched of FLUENT first version in October 1983.

Throughout several years, in May 2006, Fluent Inc. was required by ANSYS Inc. In the early1970, ANSYS was formed and evolved out of a Finite Element Analysis (FEA) modeling background and mid 1990's ANSYS was going public on the commercial market (NASDAQ). Fluent Inc. proved that ANSYS and FLUENT, both have the same vision in simulation driven product development within CAE software. Then, ANSYS was implemented to the continued advancement of Fluent Inc. to deliver the capabilities without borders with tools and functionalities to commit task easier and faster compared to before.

2.4.2 ANSYS FLUENT FEATURES

2.4.2.1 EFFICIENT AND FLEXIBLE WORKFLOW DESIGN

ANSYS Fluent workbench platform are design with easy-to-use concept; every steps are arranged with numbering and at one glance, the steps how to use can be imagined. ANSYS Fluent is consolidated into the united ANSYS workbench platform, which act as fundamental for the industry's breadth and deepest suite of advanced engineering simulation technology. When ANSYS is discovered, the function and ability of ANSYS Fluent will be delivered. There is a lot of benefits by using ANSYS Fluent that are:

- ANSYS will provide work process that allows preparation of process geometry for flow analysis without making boring rework.
- Data model will continually share across physics beyond the basic fluid flow without duplications.
- A series of parametric variations in geometry, mesh and post-processing are easily defined.
- From ANSYS, certain product or process can be improving by understand about the variability and design sensitivity.
- ANSYS provide the platform that easily set up and perform with Multiphysics simulations.

2.4.2.2 MATERIAL PROPERTIES

The detailed fundamental of materials characteristics that influence of flow conditions such as pressure, temperature, velocity, give a huge impact on the accuracy of CFD predictions. ANSYS Fluent software comes with material modelling options without borders to ensure the highest fidelity solutions can be achieved.(Your & Promise, n.d.)

ANSYS Fluent provides the full packages database of material properties for a large range phase such as liquids, gasses and solids. Besides the user can take advantage to easily define any number of new material and dependencies of material properties; ANSYS Fluent also give property to allow the user to enter any algebraic expressions for such custom models directly.

2.4.2.4 POST-PROCESSING & DATA EXPORT

The graphics, animations and reports of the CFD result on the geometry design can easily being generated by post-processing tools. ANSYS Fluent was provided with a lot of post-processing features such as shaded and transparent surfaces, path lines, vector plots, contour plots, custom field variable definition and scene constructions. Despite, the result stated can be exported to ANSYS CFD-Post or CAE packages for additional analysis. Within ANSYS workbench, ANSYS Fluent solution data can be transferred to ANSYS simulation surfaces for use as pressure loads.

2.4.2.5 MESH FLEXIBILITY

ANSYS Fluent provides with a wide range features among engineering simulation software including mesh flexibility. The capability of ANSYS Fluent in mesh flexibility was undeniable; for example, the flow problems using unstructured meshes and complex geometries can be generated with ease. Supported mesh types in ANSYS Fluent are including triangular, quadrilateral, tetrahedral, hexahedral, pyramid, prism and polyhedral. Besides, the geometry not just can be created by ANSYS but also can be imported from CAD geometry and can be analyzed for CFD use in ANSYS Design Modeler and mesh it automatically or manually by ANSYS Mesh component.(Maxwell, n.d.)

2.5 OPENFOAM

OpenFOAM is open source CFD software that widely used across most areas of engineering and science organizations. OpenFOAM was developed primarily by OpenCFD Ltd since 2004; OpenFOAM has extensive range of features to solve fluid flow simulations involving chemical reactions, turbulence and heat transfer. The meaning of OpenFOAM is Open Source Field Operation and Manipulation.

The concept of OpenFOAM is using C++; C++ toolbox is used for development of customized numerical solvers and pre- or post-processing as the solutions for branch of fluid mechanics including computational fluid dynamics (CFD) problems. OpenFOAM was released with the coding as open source software under the GNU general public license. (Conditions, 2016)

Like all the engineering simulations software, OpenFOAM have advantages and disadvantages in appropriated used. The advantages in using OpenFOAM as CFD simulations software in engineering field is OpenFOAM provide fully documented source code compared to the other open source software. This will give the user opportunity to improve or fix the OpenFOAM code to be more reliable in engineering simulations software. Besides, OpenFOAM is free and no cost will be charged on user to get the license.

The disadvantages in OpenFOAM using is the programmer's guide does not provide sufficient details, it will be hard when the process of making and adding the new applications or functionality. Besides, absence of an integrated graphical user interferes; the graphical user interface is the traditional method used to transform and import inventory data.

2.5.1 OPENFOAM FEATURES

2.5.1.1 MESH GENERATION WITH THE *SNAPPYHEXMESH* UTILITY

snappyhexMesh is a utility which can generate three-dimensional meshes that contain of hexahedra (hex) and split-hexahedra (split-hex). *snappyHexMesh* was supplied OpenFOAM mesh generated utility. By analyzing geometry design on snappyHexMesh, the file then saves in Stereolithography (STL) or Wavefront Object (OBJ) format. After that, the geometry then been transferred to snappyHexMesh to been analyzed. The parameter inserting, cutting plane, boundary condition as the process to be filled before the meshing analysis was running. (Functionality, 2016)

2.5.1.2 POST-PROCESSING COMMAND LINE INTERFACE (CLI)

In this sub-topic will describe about the command in OpenFOAM that have been unified within a single command line interface (CLI). The purpose of post-processing includes data processing sampling such as visualization, case control and run-time. The function of post-processing in OpenFOAM software can be implemented by convectional post-processing; convectional post-processing is a processing of data activity that happens after running a simulation. Next, the other function of post-processing use OpenFOAM is run-time processing; describe as data processing that is perform during the simulations has run. From this step, the flow rate calculation can be determined. The OpenFOAM provide a lot of extension that allow the function of open source become huge. From this method, the forces and force coefficient also can be determined. (Ambrosino & Funel, 2006)

2.5.1.3 TRANSPORT/RHEOLOGY MODELS

In OpenFOAM, a library of models for viscosity is included while the solver for energy and heat does not include. There is a lot of model can be created; for example, is Netonian model, Bird-Carreau model, Cross Power Law model, Power Law model, Casson model, and Herschel-Bulkley model.

Based on this research, the model related is Bird-Carreau model. This model is suitable to create model about transportation. Bird-Carreau model was involved coding that need to insert the parameter from the real model of transportation; for example, the value of speed and power of the vehicles.

2.6 GMSH

Gmsh is one of the finite element mesh generator that released under the GNU General Public License. Gmsh is an open-source software and it is free and can be downloaded directly on the its own website. Gmsh contains four modules that is for geometry, meshing, solving and post-processing. Gmsh also has provide functions that can be inserted the parametric input, and has advanced visualization mechanisms. The features of Gmsh is limited but this open-source software provides the extension-friendly that allow another extension to be used to present the result and another simulation needed. The example of the extension software is:

1. Gmsh uses OpenCascade for constructive geometry features.
2. Gmsh interfaces the following additional external mesh generators: OPenFOAM, Netgen from Joachim Schöberl and TetGen from Hang Si.
3. Gmsh's high quality vector PostScript, PDF and SVG output is produced by GL2PS.
4. Gmsh's cross-platform graphical user interface is based on FLTK and OpenGL.
5. Gmsh implements a ONELAB server to drive external solvers, as for example the open source finite solver GetDP.
6. Gmsh and GetDP are bundled in the Onelab/Mobile app for iPhone, iPad and Android devices.
7. We organized the fifth Tetrahedron workshop in Liège on July 4-5 2016.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter will describe about procedure or steps that being taken along this research. In this topic, steps in using ANSYS-Fluent and OpenFOAM are shown with details to introduce and give an idea about how this software is reacting.

The methodology of this study is summarized in the flow chart as shown in the Figure 3-1. This flow chart describes overall of the method use for both software's. First step, the geometry of model design must be created. Next, the geometry created will transfer to different software that is ANSYS-Fluent and OpenFOAM to be analyze for mesh and CFD analysis. Then the result will be shown and discuss in next chapter.

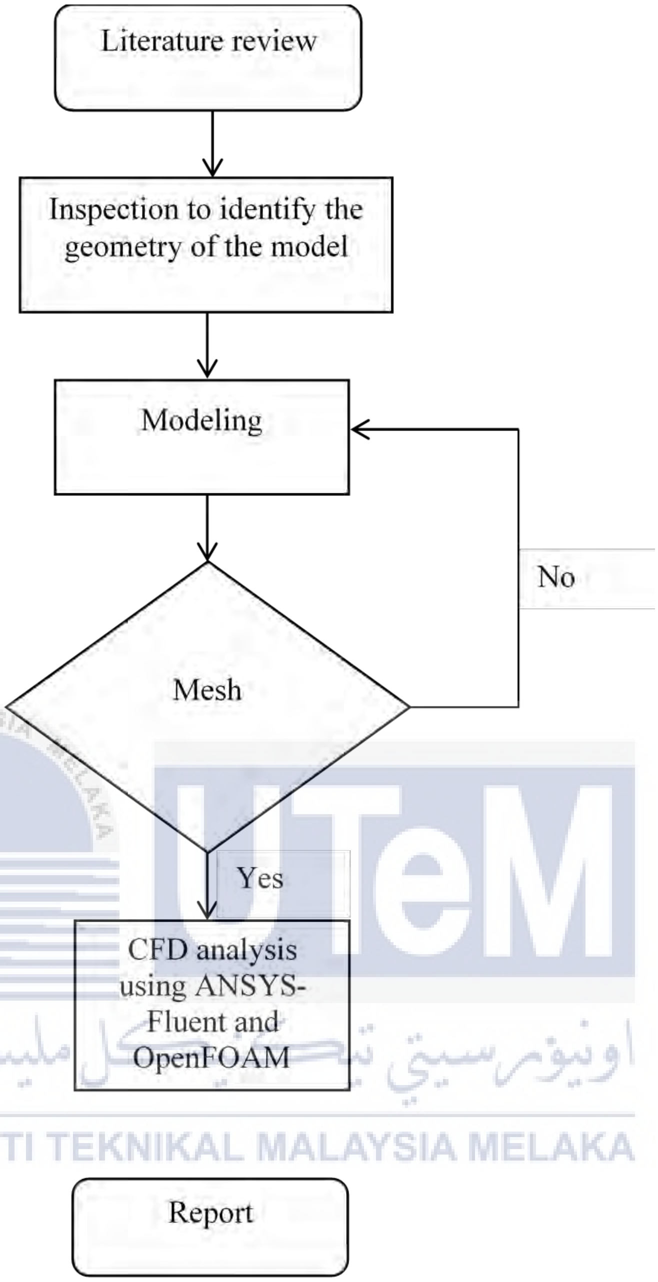


Figure 3-1 Flow chart along the research

3.2 ANSYS-FLUENT

In this sub-topic, the design of the broad physical modeling capabilities needed to be predicted. The design of the product then is tested in ANSYS-Fluent to study the impact of every fluid behavior on the product. The assumption of parameter used along this experiment in this experiment involving ANSYS-Fluent are cite from the original model.

3.2.1 INTUITIVE, PARAMETRIC & AUTOMATED WORKFLOW

The first step to be understood is about ANSYS workbench and functions of each icon's. This will be useful in ANSYS consumptions. The unique of ANSYS workbench platform can change the way engineers work with simulations. This software workbench is a little bit neat and orderly. Data record will become easier to arrange; the analysis steps are orderly arranged. The ANSYS Fluent workbench platform were categorized with labeled that easy to understand.

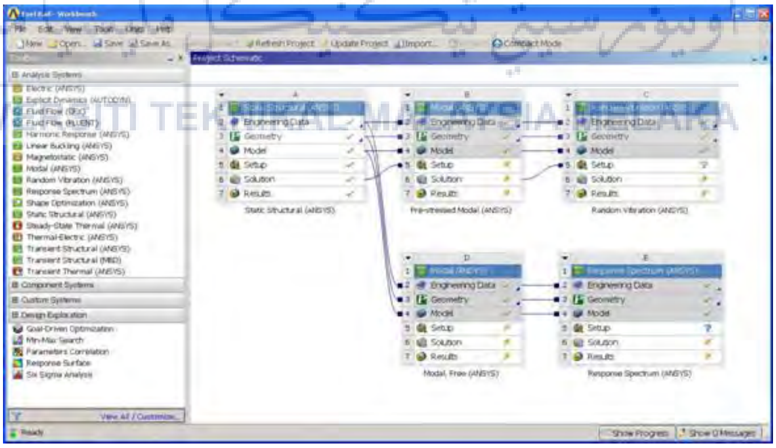


Figure 3-2 The ANSYS-Fluent workbench

3.2.2 ANSYS DESIGN MODELER (GEOMETRY)

Most of engineering simulation software needs geometry to represent the design to be simulated or tested. The design could be solid component whether for structural analysis or fluid flow analysis. ANSYS provide a system that allows design or geometry that has been produced in computer-aided-design (CAD). ANSYS design modeler allow the connection to all major CAD systems, so the existing data including parameters can smoothly being transfer to the ANSYS software. If there are corrections need to be made after the geometry were analyzed on the ANSYS, the correction can be made on CAD system. The parameter can be adjusted and the design will be updated; any feature removal and simplification is maintained. So the result of the analysis will be changes and updated automatically.(Haddadi et al., 2015)

3.2.2.1 STEPS INSERTING GEOMETRY FROM CAD SYSTEM

1. The first steps to be taken is to insert the geometry design from SolidWorks is the drawing from SolidWorks need to be save as IGS file. For example, BILLET.igs.

2. The fluid flow (fluent) was chosen as component system on ANSYS-Fluent workbench; the geometry button was pushed to insert the geometry.

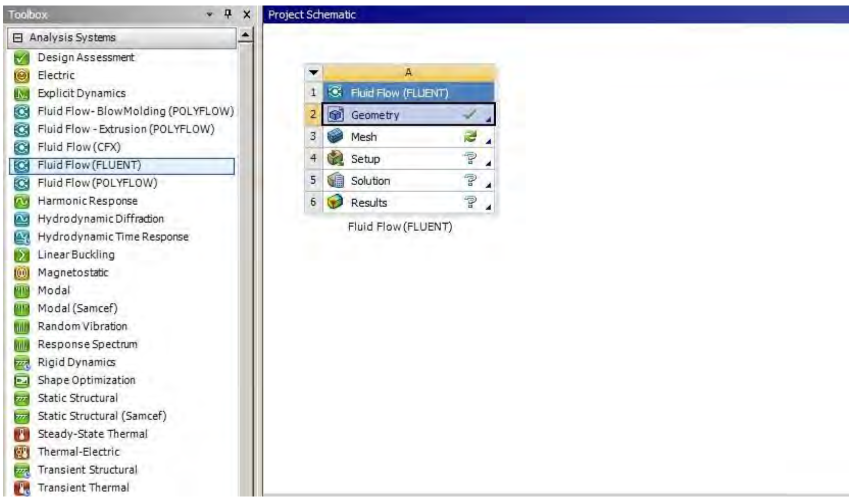


Figure 3-3 Project schematic diagram for fluid flow (fluent)

3. The geometry that was renamed as IGS file then inserted on ANSYS Fluent workbench.

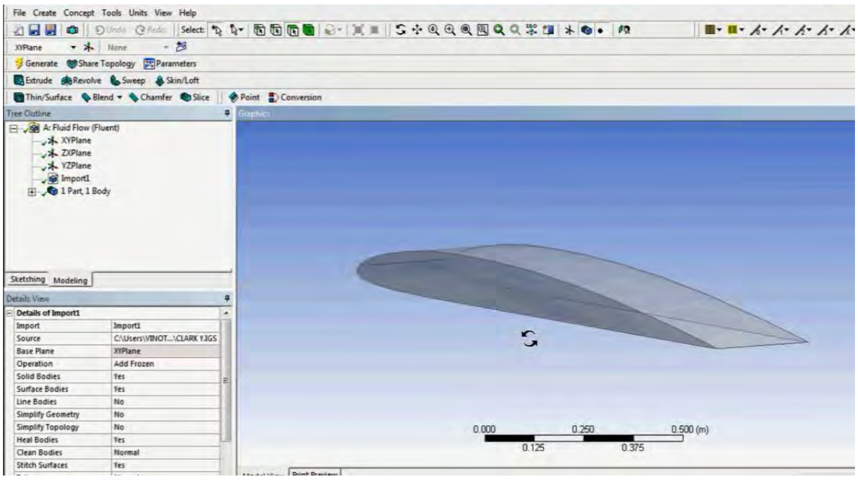


Figure 3-4 Geometry inserted on ANSYS-Fluent workbench

3.2.3 MESHING IN ANSYS-FLUENT

All engineering simulation software, meshing constitute as the initial steps in analyzing of any geometry. In ANSYS Fluent, meshing is done in ICEM CFD software. Meshing is important steps in pre-processing phase because the mesh is representing of the physics behind the real part that the model is referring to. From mesh analysis, the effectiveness and efficiency of an analysis can be determined.

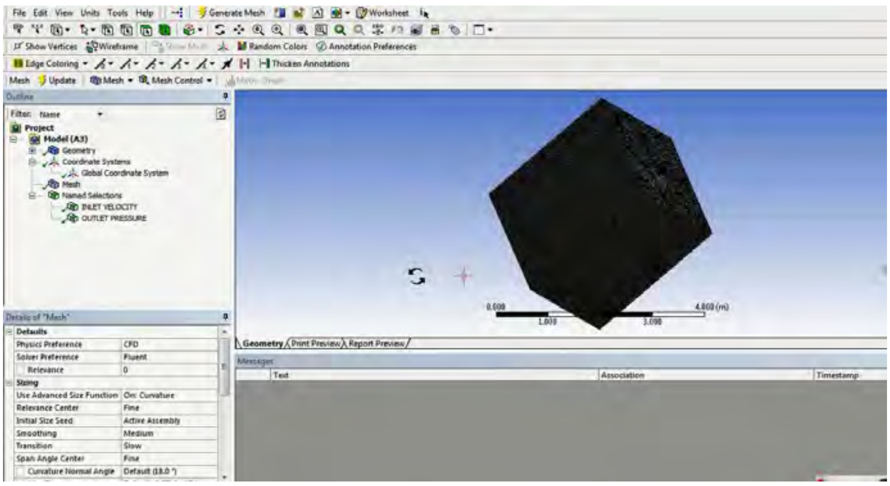


Figure 3-5 Meshing graphis by ANSYS-Fluent

3.2.4 ANALYSIS SETUP IN ANSYS-FLUENT

In this stage, analysis fluent database was setup according to data collected from the past research about model tested. The material used was for the model body was aluminum and carbon reinforced plastics that cover with paint. Then, the data about fluid acting on the geometry also has been filled. The fluid chosen is air, when the car is moving on the road, the air played as main role acting on the surface of the car body. The speed of air flow acting on the body surface of geometry also been considered. Besides, the boundary conditions of geometry were specifying for both inlet and outlet. Based on this research the inlet of the geometry is on front of the car body; because when the car is moving, the air flow must be reverse to car moving flow.

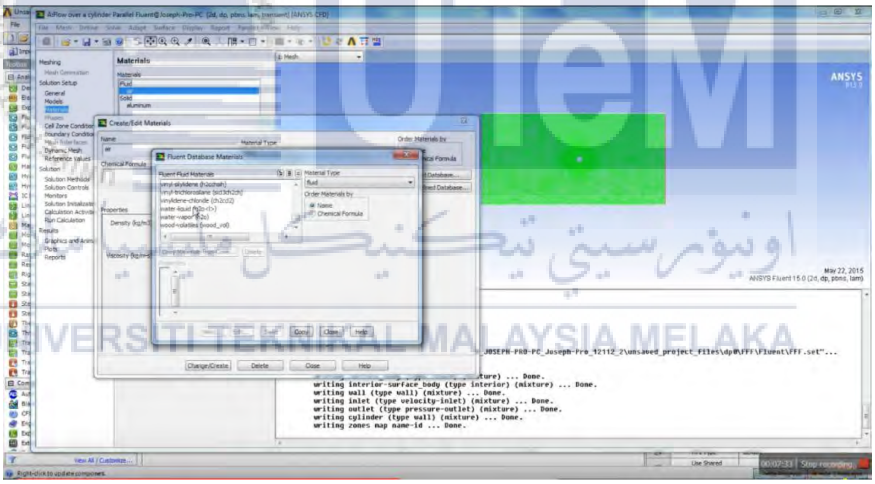


Figure 3-6 ANSYS-Fluent database material of geometry

3.2.5 SOLUTION AND RESULT ANALYSIS

On the solution sub-component, there are solutions method, solutions control, monitors, solutions initializations, calculations activity and the last is run calculations. After all the parameter for both flow air and geometry simulation flow is filled, run calculations button was pressed in order to get the result for flow analysis acting on the geometry. The result was shown in type of graph plotting. Otherwise, the full report about the geometry designation or strength also can be shown. From the full report provided by ANSYS Fluent, the problems about the product can be determine and fix before the prototype was produce.

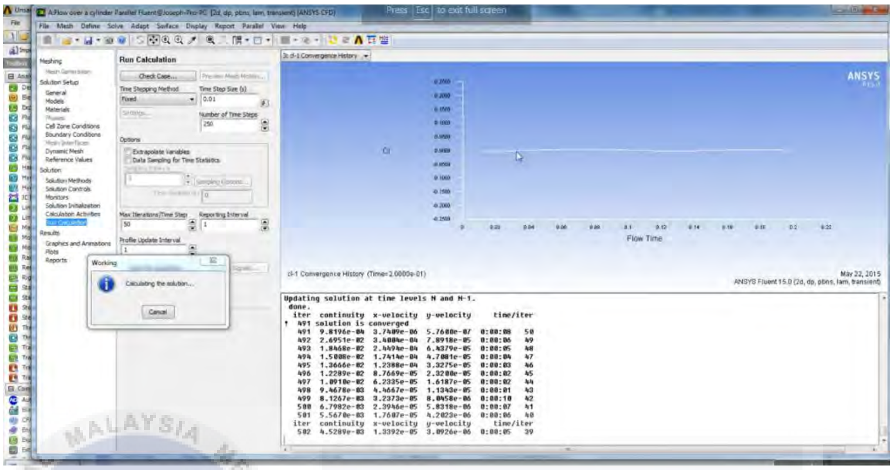


Figure 3-7 The result analysis of ANSYS-Fluent

3.3 OPENFOAM

From the past study, the parameter of specifications of aero-foil is being collected and recorded. The OpenFOAM software needs an extension to make certain analysis before the analysis drag to OpenFOAM to be analyzed overall. (Ambrosino & Funel, 2006)

3.3.1 OPENFOAM GEOMETRY MODELER

OpenFOAM allow the geometry that has been drew by SolidWorks to be transferred to the software workbench. The file of geometry design must be saving under STP file; for example, of file name is BILLET.stp. After the file name was setup in stp format, then the geometry can be inserted directly in salome OpenFOAM software.("Salome OpenFOAM Tutorial - CAD model to Solution Complete," n.d.)

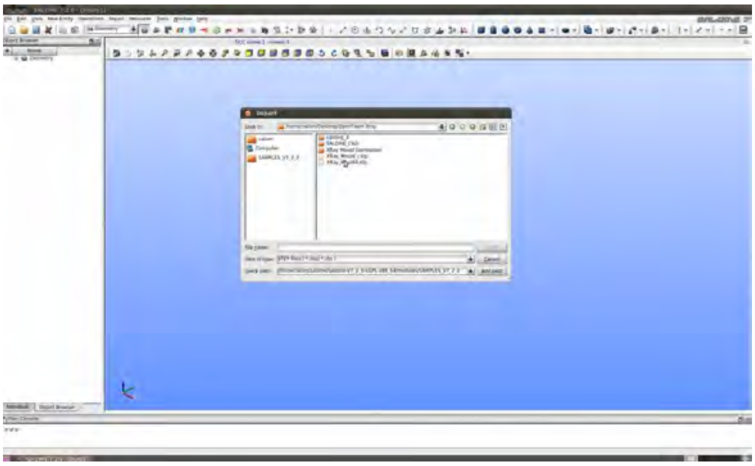


Figure 3-8 The geometry design file name

3.3.2 MESH GENERATION WITH THE SALOME OPENFOAM

This sub-topic was described about the steps that being taken while making the mesh analysis on the geometry design. OpenFOAM need extension software to made meshing analysis that be names Salome OpenFOAM. The file format of geometry is in (STP).

3.3.2.1 STEPS IN MESHING GENERATION

1. After the geometry was inserted to Salome OpenFOAM software, create group for the geometry design by filled the suitable parameter like shape type, group name, main shape, and main shape selection restriction.
2. After the group was made, create the boundary conditions of the geometry design by define the inlet and outlet of the fluid flow. For example, fluid flow in car applications, the inlet flow that is air is placed on the front of car body, while the outlet of air flow was placed at the back of car body. (Conditions, 2016)

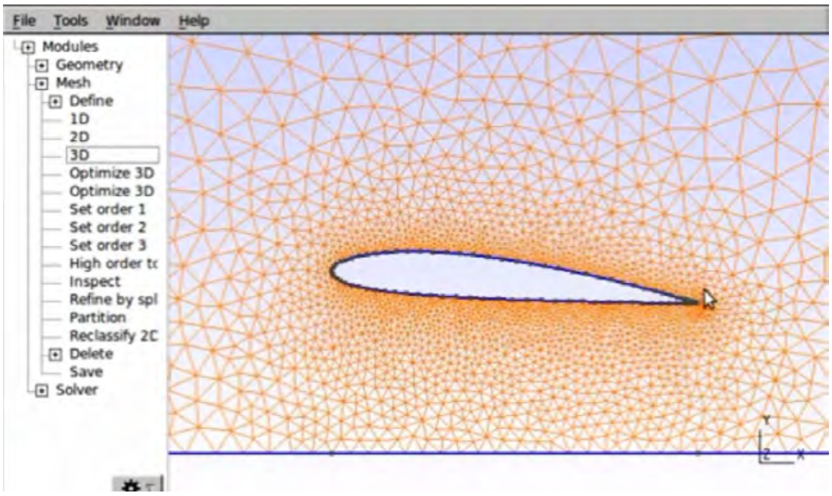


Figure 3-9 The inlet fluid flow was setup

3. Parameter of hypothesis construction was filled, then open back the ‘create group’ then select the ‘select all’ button to get the coordinate.
4. Then click the ‘calculate’ button on the run mesh and the result of the mesh on the geometry was shown.

3.3.3 PARAVIEW/PARAFOAM GRAPHICAL USER INTERFACE (GUI)

1. The main post-processing tools provided with OpenFoam are a reader module to run with a visualization application that is paraView. paraFoam is an intermediary to launch paraView using the module reader provide by OpenFOAM. This extension was executed like any of OpenFOAM extension utility that cab be executed by single command.

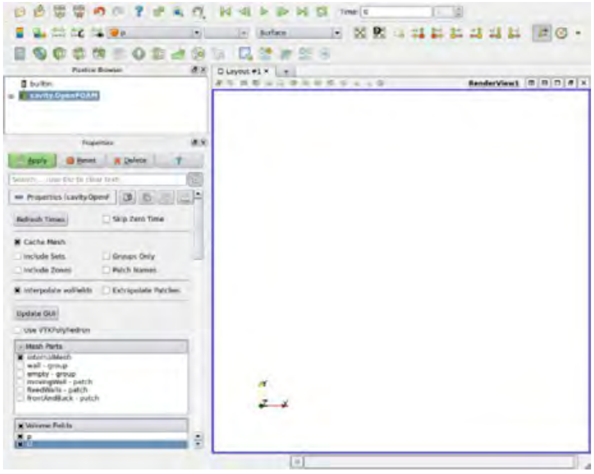


Figure 3-10 The example of OpenFOAM utility

2. The parameter of geometry design was inserted by referring the original size models. The parameters panel that contains the setting for mesh, field and global controls were includes at the properties windows at the module.

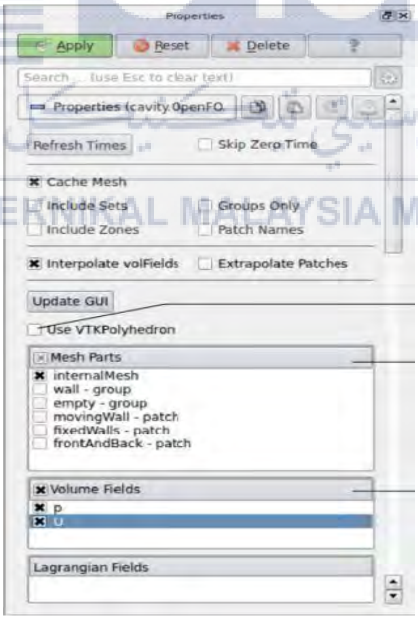


Figure 3-11 The properties window of parameter panel

3. Run the systems of geometry design setting with parameter as the original model. Then the result of flow fluid contour is shown on the contour plots. Next, the streamlines are creating on the tracer lines using the Stream Tracer filter. The streamlines data then being launched and the background color are changes in order to see the result clearly. After all the analysis are finished been setting, display the image and animation output.

Figure 3-12 The airflow direction of streamline in OpenFOAM

3.4 GMSH

The Gmsh is one of the finite mesh generator that provide Computational Fluid Dynamics (CFD) functions. The CFD features that supported by this software are geometry modelling, meshing and post-processing. The features provide by Gmsh is limited but Gmsh also offers extension-friendly that give permission of Gmsh to connect with the other software to make the other simulation analysis.

3.4.1 GEOMETRY MODELLING FOR GMSH

The Gmsh provides the features that allow this software to create its own geometry modelling. It is means that the geometry modelling can be create directly on this software. But in this project, the geometry has been import from the SolidWorks. The geometry then being analyze in the meshing analysis.

3.4.2 MESHING IN GMSH

The geometry of the aero-foil model that has been modified in the Gmsh then has been saved with the name.geo. On the Gmsh software, the meshing has been created by selecting the 3D model from the mesh menu. In the meshing process, the size of the mesh element can be controlled by setting the Element Size Factor under the mesh in the option windows. The smaller the number of the size factor, the smaller the mesh element.

Figure 3-13 The meshing in GMSH software

3.4.3 AIRFLOW ANALYSIS OF GMSH

Due to the limitations of the Gmsh features in CFD analysis, the airflow analysis of the Gmsh has been conducted using the OpenFOAM software. The Gmsh software is providing the airflow functions of features but it is limited and to provide more functions, it needs another extension software to represent the result. The airflow steps of the model then been conducted as in the OpenFOAM software.

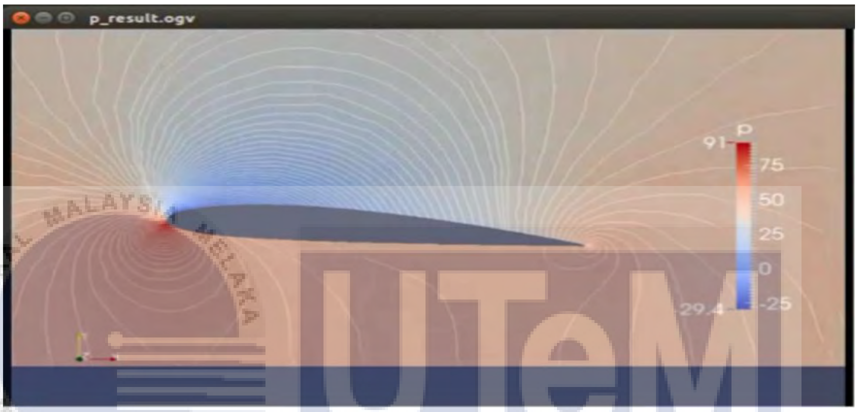


Figure 3-14 The airflow analysis in OpwnFOAM

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

The experiment and all the data about the research will be presented in this chapter. The visual about the flow direction of the commercial package and open-source software are expose. The result that provide from the software testing will be presented in this chapter in order to understand and study about the flow and meshing to get the smooth result with less error.

4.2 MODEL

The model that has been chosen to be testing in CFD analysis simulations is a cross-sectional of aero-foil model. The concept of aero-foil model is aerodynamics that allows the airflow to go through and make a contact surface with the model. This model has been chosen because of this model play the important role in the industry field; there are a lot of researcher are studies about this model because this model is using in aero-plane. Aero-plane is a vehicle that carries passenger and being fly in the sky, so the simulations and all the criteria of the model is important and should be conduct with care because it will affect the passenger. The reasons why this model being chosen is also because of this model is popular among the researcher and the concept used in making this design is difficult.

4.3 CAE TOOLS IN ANSYS-FLUENT, SOLIDWORKS, OPENFOAM AND GMSH

Computational-Aided Engineering (CAE) tools are popular in market simulations, as the software that support CAE functioning is divided into two categories that is commercial-package software and open-source software. the type of fields and phases of CAE tools are Finite Element Analysis (FEA) that combination of pre-processing and post-processing, Computational Fluid Dynamics (CFD), MultiBody Dynamics (MBD) and kinematics, Analysis tools for process simulation for operations, Optimization, Meshing, and Analysis solver.

Table 4-1 CAE tools in ANSYS-Fluent, SolidWorks, OpenFOAM and GMSH

CAE fields and phases	Commercial-Package software		Open-source software	
	ANSYS-Fluent	SolidWorks	OpenFOAM	Gmsh
Finite Element Analysis (FEA) (pre-processing and post-processing)	√	√	√	√
Computational fluid dynamics (CFD)	√	√	√	√
Multibody dynamics (MBD) and Kinematics	√	√	X	X
Analysis tools for process simulation for operations (E.g.: casting and molding)	X	X	X	X
Optimization	√	√	X	X
Meshing	√	√	√	√
Analysis solver	√	√	√	√

The table shows about the comparison of each software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh, tha has been chosen to be used along the research in terms of CAE tools applications. From the results recorded and be fill in the table form shows that the four-software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh, providing the CAE tools of Finite Element Analysis (FEA) and also Computational Fluid Dynamics (CFD). While for MultiBody Dynamics (MBD) and kinematics, just ANSYS-Fluent and SolidWorks that provide that functions of CAE tools applications. Next, CAE tools applications is analysis tools for process simulation for operation such as casting and molding, all the four-software are not provides and support the tools. Besides, for optimization just ANSYS-Fluent and SolidWorks that provide that function of CAE tools. Although, all the four-software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh, are provides the function of meshing and analysis solver.

Finite Element Analysis (FEA) is one of the CAE tools applications that provides a function to test the strength of model. There are a lot of functionality in FEA on CAE fields, such as structural analysis, heat transfer, fluid flow, mass transport and electromagnetic potential. All the four-software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh, provides this function means that, all these software are worthy to use to test for the FEA testing. This also can be conclude that, the open-source software also provides the function same as the commercial package.

Next is Computational Fluid Dynamics (CFD); CFD applications has been chosen to be analyze along this reseach. The function of CFD applications is CFD provide some feature that can shown the user about the flow simulations also the part of maximum and minimum that attact the surface. All the four-software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh, provides this feature and making all the four-software on the same level wether one catogories for open-source and another categories is on commercial-package.

For MultiBody Dynamics (MBD) and kinematics are providing by ANSYS-Fluent and SolidWorks, while the other software are not supported with this features. This will means that the commercial-package software get advantage for this feature compare to open-source software that didn't supported this feature and need to use other software that provide this analysis. MultiBody Dynamics (MBD) and kinematics system is one of the solid bodies or link that connected to each other by joining with the restrict in its relative motion. (Corporation, 2011)

Optimizations is one of the applications in CAE tools that provide a function to optimize the model without repeating all the steps from the start. Optimizations are using for optimize the critical part from the model and inserting and replacing the new value to provide the new result. The software that support the functionality is just from commercial-package that is ANSYS-Fluent and SolidWorks while for open-source software, didn't provides this features.

Meshing and analysis solver is one of the features that connected with each other. Analysis solver is the applications that the steps need is to setup the boundary parameter that is inlet and outlet of the flow. While, for meshing,; meshing is one of the applications that provide the visualizations about the critical part of the model.All four-software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh, are provides this features. This will be advantages for open-source software that shows that open-source software can be functioning likes commercial-package.

4.4 COMPARISON IN CFD

Table 4-2 Comparison for CFD features among the ANSYS-Fluent, SOlidWorks, OpenFOAM and GMSH (Corporations, 2017)

CFD features	Commercial-Package software		Open-source software	
	ANSYS-Fluent	SolidWorks	OpenFOAM	Gmsh
Liquid and gas flow with heat transfer	√	√	√	√
External and internal fluid flows	√	√	√	√
Laminar, turbulent, and transitional flows	√	√	X	X
Time-dependent flow	X	√	X	X
Subsonic, transonic, and supersonic regimes	X	√	X	X
Gas mixture, liquid mixture	√	√	X	√
Conjugate heat transfer	√	√	√	X
Heat transfer in solids	√	√	√	X
Incompressible and compressible liquid	√	√	√	X
Compressible gas	√	√	√	X
Real gases	X	√	X	X
Water vapor (steam)	X	√	X	X

The table above shows that the listing of CFD features among the four-software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh. Based on the table above, the best software that should be use in the industry field is SolidWorks. This is because from the list above, SolidWorks provides all the features above compare to ANSYS-Fluent, OpenFOAM and Gmsh. Compare to the open-source software (OpenFOAM and Gmsh), the commercial-package software (ANSYS-Fluent and SolidWorks) provide more features in CFD simulations analysis. But, open-source has its owh benefit, it has an open license that give permission to get the connection with the other extension software. For example, Gmsh want to make a meshing, the result of the meshing in Gmsh can be drag to the OpenFOAM and the OpenFOAM can present the result.

4.5 MESHING

Meshing is a geometric domain that will be practice of generating a polygonal or polyhedral mesh. Usually, meshing is used for version to a computer screen or for physical simulation such as Finite Element Analysis (FEA) and Computational Fluids Dynamics (CFD). The input model or rear design form can be created on 3D model software like CAD and SolidWorks. This meshing is an analysis that have contributions of mathematics, computer science, and engineering field. Meshing analysis is divided into four categories that is tetrahedral, pyramids, prism and hexahedra; this four category is usually used in three-dimensional meshed. Commonly, arbitrary polyhedral meshed is used for Finite Element Analysis (FEA). Generally, for Finite Element Analysis need to consist of piecewise structured arrays of hexahedra known as multi-block structured meshes.

4.5.1 MESHING IN ANSYS-FLUENT

As the other engineering that focus on Computational Fluid Dynamics, ANSYS-Fluent also need go through meshing analysis before the other analysis have done. In ANSYS-Fluent, meshing is a common thing that can demote the pre-processing phase of the Finite Element Analysis (FEA) which uses the Finite Element Method (FEM).

ANSYS meshing is a general-purpose, intellectual, mechanical high-performance product. Meshing in ANSYS-Fluent provide accurate, efficient, and metaphysics solutions. Besides, just in one click, a well suited mesh for a specific analysis can be generated for all parts in a model or design. ANSYS-Fluent also provide full controls over the options used in order to easier the expert to create and fine-tune the analysis. The special of ANSYS meshing is the power of handling is automatically used to reduce time for mesh generation.

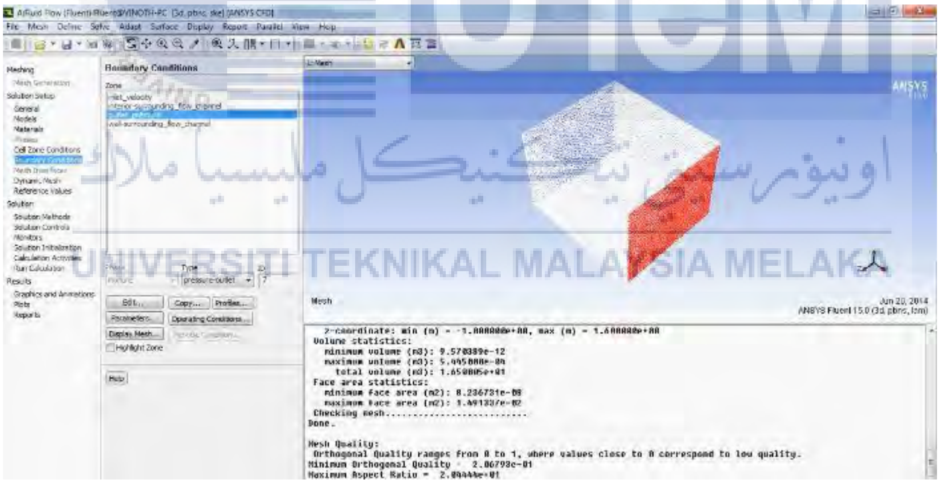


Figure 4-1 Meshing in ANSYS-Fluent

4.5.2 MESHING IN SOLIDWORKS

Finite Element Analysis (FEA) provides a reliable numerical technique for analyzing engineering designs. The analysis is started from the subdivided the 3D model into small pieces of simple shapes or elements connected at common points that is known as nodes. Meshing is important to analyze the most risk part that can broke the product.

The meshing that SolidWorks software provided is 3D tetrahedral solid elements, 2D triangular shell elements, and 1D beam elements.

SolidWorks provides the automatic meshed in the software based on a global element size, tolerance, and local mesh control specifications. The controller that have been setting in this software make it easier to specify the different sizes of elements components, faces, edges, and vertices. The function of global element size is to consider the volume, surface area, and other geometric details. The result generated mesh is depending on the geometry and dimensions of the model, element size, mesh tolerance, mesh control, and contact specifications. The small element is requiring to get accurate result.

4.5.3 MESHING IN OPENFOAM

OpenFOAM are provided with *polyMesh;* and *polyMesh* are divided into two categories that are *snappyHexMesh*, and *blockMesh.* These two types of mesh are a specifications of the way the OpenFOAM classes handle a mesh. In this research, the mesh that have been used is *snappyHexMesh.*

This mesh offer 3D meshes covering hexahedra and split-hexahedra automatically from triangulated surface geometry on the model. The mesh design is approximately follows the surface by starting the mesh and altering the resulting split-hex mesh to the surface. While the other surface phase will shrink back the resulting mesh area and start to showing the insert cell layer. In order to make the meshing is more quality; the specifications of mesh refinement level are handle the surface forcefully and stretchy. OpenFOAM run parallel with a load balancing step every iteration.

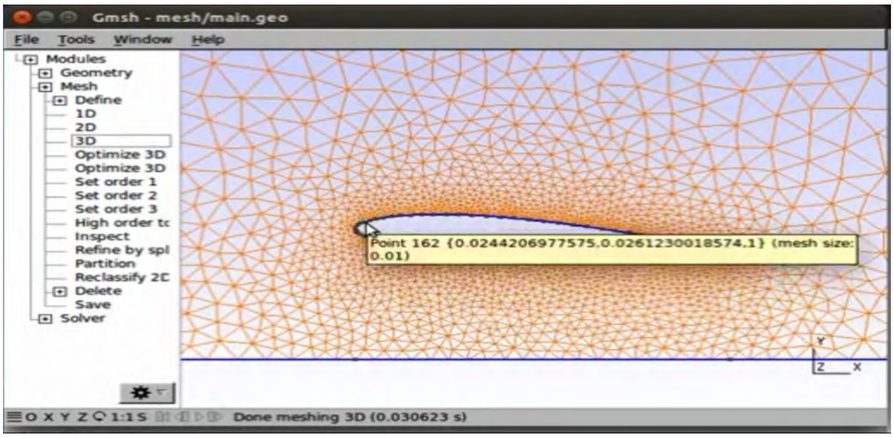
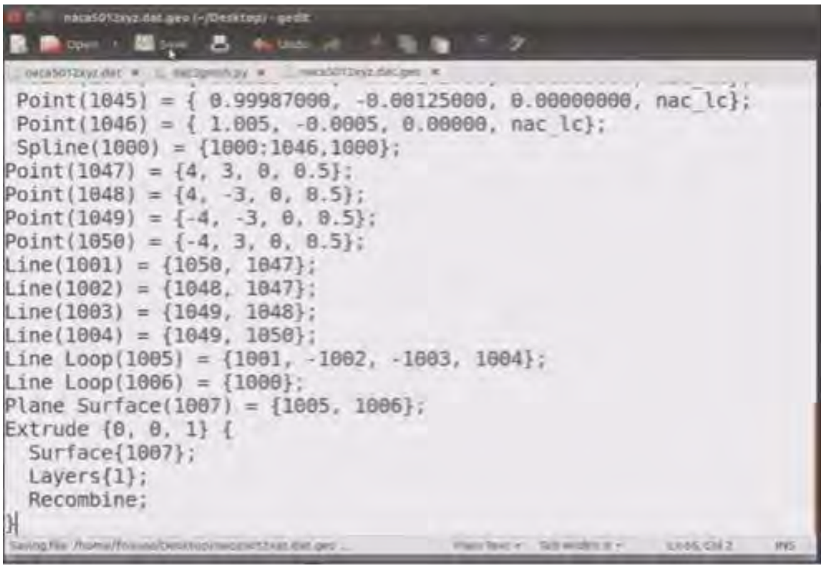


Figure 4-2 Meshing in OpenFOAM

4.5.4 MESHING IN GMSH

Gmsh is a three-dimensional Finite Element Analysis (FEA) mesh generator. This software provides with built-in pre- and post- processing facilities. The specifications of Gmsh is this software is design as a free 3D finite element grid generator that built- in with CAD engine and post-processing. Usually, Gmsh open-source software is using to analyze the 2D design. This simulation software can be combine with OpenFOAM to create the other analysis.

Same as most of the open-source software, every command is in coding code. Gmsh mesh is a little bit difficult to handle affect from the coding needed to enter. The details and all the parameter needed to insert in Gmsh as a coding.



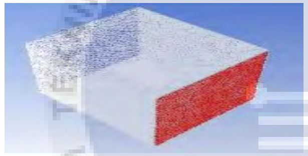
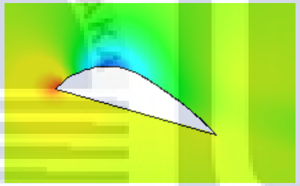
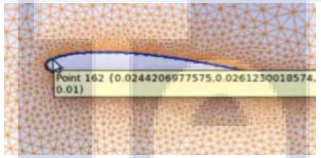
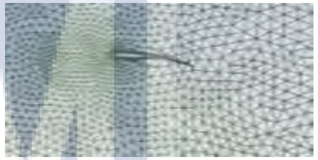
```
 Point(1045) = { 0.99987000, -0.00125000, 0.00000000, nac_lc};
 Point(1046) = { 1.005, -0.0005, 0.00000, nac_lc};
 Spline(1000) = {1000:1046,1000};
Point(1047) = {4, 3, 0, 0.5};
Point(1048) = {4, -3, 0, 0.5};
Point(1049) = {-4, -3, 0, 0.5};
Point(1050) = {-4, 3, 0, 0.5};
Line(1001) = {1050, 1047};
Line(1002) = {1048, 1047};
Line(1003) = {1049, 1048};
Line(1004) = {1049, 1050};
Line Loop(1005) = {1001, -1002, -1003, 1004};
Line Loop(1006) = {1000};
Plane Surface(1007) = {1005, 1006};
Extrude {0, 0, 1} {
  Surface{1007};
  Layers{1};
  Recombine;
}
```

Figure 4-3 Coding of the mesh in GMSH

Figure 4-4 Mesh analysis in GMSH

4.6 COMPARISON OF SOFTWARE IN TERMS OF MESHING

Table 4-3 COmparison of software in ter of meshing

Type	ANSYS-Fluent (commercial-package)	SolidWorks (commercial-package)	OpenFOAM (open-source)	Gmsh (open-source)
Meshing				

Based on the result obtain in table 4-3 , the all four-software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh shows the different type of meshing visualizations and each of the concept give the different meaning. For the ANSYS-Fluent, the meshing is done in type of cube, means all the meshing is done on the surrounding of the model. The boundary must be inserted as the paramater of the model. The boundary set is inlet of the flow and outlet of the flow.

Next, for the OpenFOAM software, the meshing has been done and the surface that been meshing is directly around the model. In OpenFOAM also need to insert the boundary conditions of parameter like inlet and outlet of the flow. The meshing has been done along the surface of the model itself and the citical part are shown in term of triangulated surface geometry being more crowded on the model. While for the Gmsh, the meshing process can be done by this software itself and it can be done the meshing process on the OpenFOAM as the extension software. The function of the meshing is same like the OpenFOAM, the visualizations is by the triangulated surface geometry and the critical part will be shown as the crowded the triangulated surface geometry it is.

In terms of the detail of the meshing design, the commercial software tools have a lot more user friendly design as the ratio for triangulate surface design in the model is smaller. This result determine the complexity of the design can be done smoother and the visualization of model is clearer. The presentation of the model meshing also better with different colour on the meshing which differentiate the condition on the model. While for open-source tools both share the same bad visualization of meshing as the mesh are rougher as can be seen by naked eye. The visualization also looks too simple which make the type of data that can be obtain from this mesh is limited. The mesh also use a monotone colour for the meshing which can confuse the user to find the condition differences in the meshing.

In terms of operational process to create the meshing, the commercial software has a better control which each command is assign to a button and has their own icon. While for the open-source software, the command control is done manually by creating a coding as a command process to create the meshing which make the process more difficult.

4.7 AIRFLOW DIRECTION

Flow direction simulation is a simulator for visualizing airflow around object. This is the initiative user to understand the concept and restrict of the flow direction. An intuitive the user interferes and interoperability with the flow direction simulation is because to gain the better insight earlier in the development process. This steps is to encourage the user to lower the cost of the development.

4.7.1 AIRFLOW DIRECTION IN ANSYS-FLUENT

The airflow simulations enable the user to explore the how the design can be interacted with the wind. From this airflow direction simulation, the visualization for where the weakness, high- and low- pressure regions will form.

The velocity of airflow is considering as 133 m/s. The force acting on the x-direction is 68.93684 N; while the lift force along the y-direction is 111.76134 N. The length of the model that is wing is making short in order to make the short and quick analysis. Hence, the lifting value that have been obtained is a little bit low.

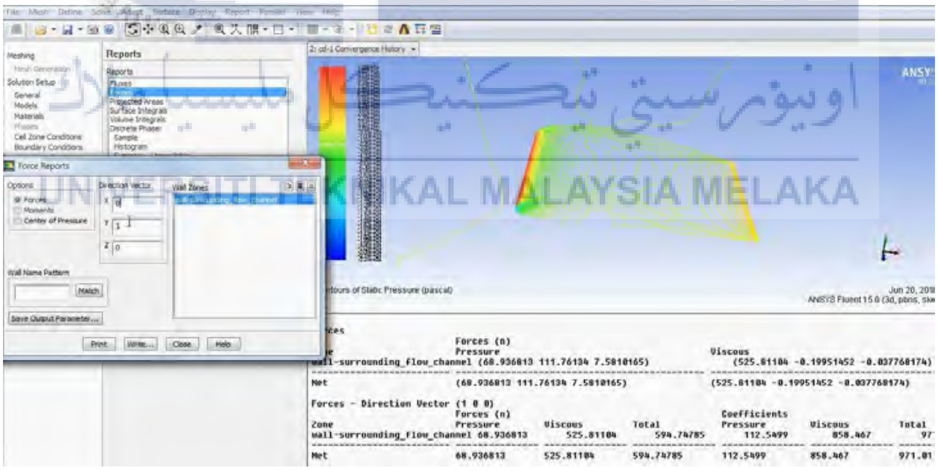


Figure 4-5 Airflow direction and x-direction force on ANSYS-Fluent

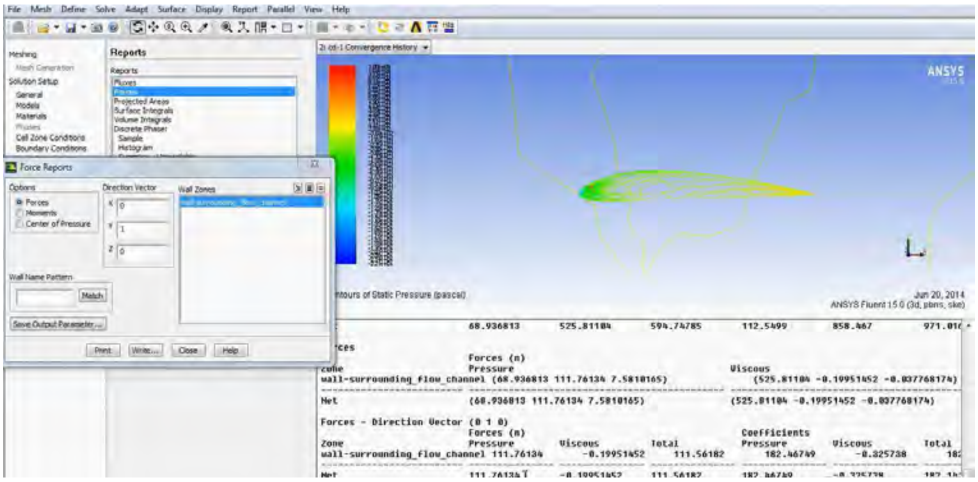


Figure 4-6 Airflow direction and y-direction force on ANSYS-Fluent

4.7.2 AIRFLOW DIRECTION IN SOLIDWORKS

Air-foil analysis is an analysis that allow user to see the direction of the flow acting on the model. Airflow direction analysis is important, especially for airfoil. It is because from the analysis, the weaknesses of the product can be identified.

In this research, the result obtains in the type of flow. The parameter that included in software is velocity; velocity was remain constant for all four software that is 133 m/s. From the result the highest temperature that the model can overcome is 296.24 K and the lowest temperature is 289.48 K. Besides that, the another analysis that can be analyze from this software is pressure that the model can overwhelmed; the highest pressure that the model can hold is 105343.59 Pa and the lowest pressure is 96957.68 Pa. The Value can be identified as the lowest and the highest by seeing the color on the workbench value. The red color means the highest value or the critical value and the blue one for the lowest and the minimum value that the model can attend.

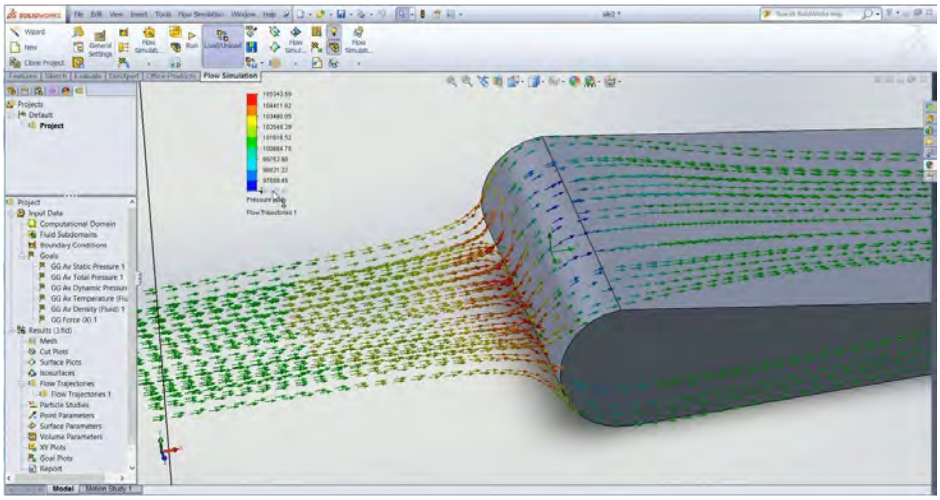


Figure 4-7 The analysis of pressure flow

4.7.3 AIRFLOW DIRECTION IN OPENFOAM AND GMSH

For the open-source software, the function is limited. That means the flow analysis that the OpenFOAM software provide also limited. The flow analysis that this software offer is incompressible flow, compressible flow, multiphase flow, heat transfer and buoyancy-driven flows, and particle-tracking flows. While for Gmsh, the flow analysis for this software is same like OpenFOAM software. This is because the flow analysis has done in the same software that is OpenFOAM. The coding of mesh of Gmsh then been substitute to the OpenFOAM software to be analyzed. The analysis for both OpenFOAM and Gmsh have the same reading and parameter.

This analysis used constant velocity that is 133 m/s for all the four software. The other parameter is just setting the boundary of the model. Number of iterations of this analysis is 60. After all the data boundary have been filled, the model starts to be analyze. The result of this airflow direction simulations will be shown as figure below.

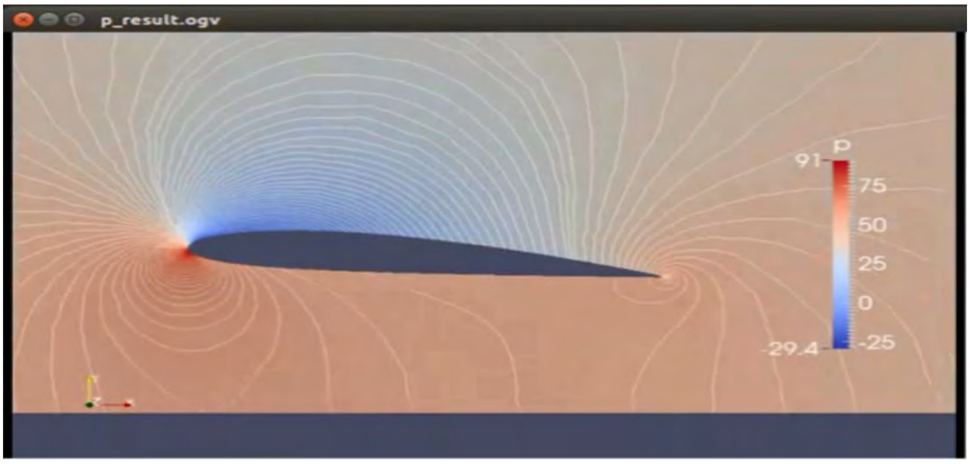
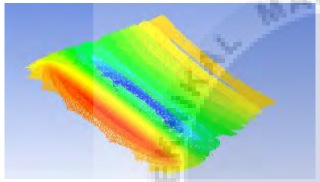
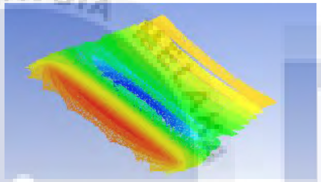
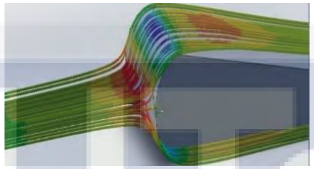
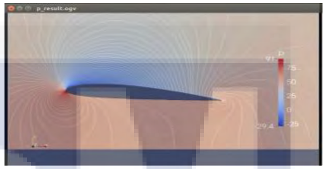
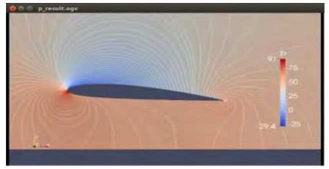


Figure 4-8 Airflow analysis for both OpenFOAM and GMSH

4.8 COMPARISON OF SOFTWARE IN TERMS OF COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

Table 4-4Comparison of four-software in tems of Computational Fluid Dynamics (CFD) analysis

Airflow Visualizations	ANSYS-Fluent (commercial-package) 	ANSYS-Fluent (commercial-package) 	SolidWorks (commercial-package) 	OpenFOAM (open-source) 	Gmsh (open-source) 
Number of iterations	60	60	60	60	60
Velocity (m/s)	133	133	133	133	133
Pressure flow	1. Maximum value $= 25.9\ kPa$ 2. Minimum value $= -64.5\ kPa$	1. Maximum value $= 26.7\ kPa$ 2. Minimum value $= -62.9\ kPa$	1. Maximum value $= 25.6\ kPa$ 2. Minimum value $= -65.3\ kPa$	1. Maximum value $= 91.0\ kPa$ 2. Minimum value $= -29.4\ kPa$	1. Maximum value $= 91.6\ kPa$ 2. Minimum value $= -28.9\ kPa$
Temperature flow	1. Maximum value $= 294.87\ K$ 2. Minimum value $=287.85\ K$	1. Maximum value $= 290.45\ K$ 2. Minimum value $=285.42\ K$	1. Maximum value $= 296.24\ K$ 2. Minimum value $=289.48\ K$	-	-

The table above shows that the comparison of four-software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh, in tems of Computational Fluid Dynamics (CFD) analysis with the previous study. The CFD features that has been chosen to make a comparison for all the software is in term of airflow visualizations, number of iterations, velocity, pressure flow and temperature flow. All the software are being compare to the previous study that using ANSYS-Fluent are because of to see clearly which of the software are more precise.

The number of iterations and the velocity of the flow simulations are being choose and fixed like the past-study data. The data for the iterations is 60 and the velocity of the flow directions is 133 m/s. The velocity chosen is due to the standard speed of the aeroplane that using the concept of aerodynamic on the aerofoil.

In term of airflow visualizations, all four-software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh, has precise visualizations and making the user understand how the flow directions are acting on the model surface. Compare to the commercial package software, the open-source software data is a little bit unprecisely; this is because, the airflow simulations are in monotone and the critical part of the simulations cannot be identified. While for commercial package, the airflow directions visualiztions are precise, the direction is stated and the variations of the color gave the explainations about the crtical part and minimum part that less react to the airflow. This can be concluded that the commercial-package software are more precise compared to the open-source software.

Next, the comparison is making in terms of pressure flow. All the four-software, ANSYS-Fluent, SolidWorks, OpenFOAM and Gmsh, has been compare to the past-study data and being compared. From the table above, the clear comparison in term of the precise of the data compared to past-study (Swanson, 2014) (Maximum value,$25.9\ kPa$ and minimum value,$-64.5\ kPa$) data can be ranked as SolidWorks (Maximum value,$25.6\ kPa$ and Minimum value,$-65.3\ kPa$), ANSYS-Fluent (Maximum value,$26.7\ kPa$ and minimum value,$-62.9\ kPa$), Gmsh (Maximum value,$91.6\ kPa$ and minimum value,$-28.9\ kPa$) and OpenFOAM (Maximum value,$91.0\ kPa$ and minimum value,$-29.4\ kPa$). As the table shows the result of the four-software conpare to the past-study, the SolidWorks data is more accurate and followed by ANSYS-Fluent, Gmsh and OpenFOAM. The different data getting from the past-study (ANSYS-Fluent) and the commercial-package ANSYS-Fluent is because of the error during making the simulation testing on the model. The precautions that can be made to make the data more precise is using the same model and meshing analysis as the past-study.

The last comparison is about the temperature flow among the model surrounding. The table shows that the software that can be compared to the past-study (Maximum value,$294.87\ K$ and minimum value,$287.85\ K$) is just ANSYS-Fluent (Maximum value,$290.45\ K$ and minimum value,$285.42\ K$) and SolidWorks (Maximum value,$290.45\ K$ and minimum value,$285.42\ K$), due to the limittaions of the open-source features on the simulations. The open-source software that are OpenFOAM and Gmsh does not support the temperature flow simulation features. This will be the disadvantges and the limittaions of the open-source software tools. Open-source software is provides the temperature flow simulations but the features is available on the other open-source software.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 INTRODUCTION

From the understanding of the report studied, this chapter will be describing about overall of this project. The conclusion that have been made can be concluded and explained from the studying of this report along the result and analysis information given among the report. This chapter also stated about some of the recommendation in order to make the system or software more useful in research on this project.

5.2 CONCLUSION

The aerodynamics model has been chosen in this project is because of the easy design of the model. The model of the aerodynamics that has been used in this project is not the overall part but it is the cross-sectional area or half of the real part. This project is about the using of Open-source software and the commercial software on the computational fluid dynamics (CFD) simulations. The open-source software that been chosen are OpenFOAM and Gmsh, while for commercial package software are ANSYS-Fluent and SolidWorks. Therefore, the main reason of this project is to investigate the differential between the open-source software and commercial package software.

Besides, to achieve the objective of this project, the analysis and intentional about the procedure to use the software are being studied and understand. The result and discussion has been stated in the chapter 4 before. From the analysis, the conclusion can be made as commercial package software and open-source software have its own pros and cons, advantage and disadvantage.

The open-source software can be stated as easy to be downloaded and it is free to use and it will be benefit in using open-source software. This open-source software can be manipulating or adjusting the function by professional. It is also user-friendly, means that this open-source software can be using with other extension-software; for example, for meshing Gmsh can used OpenFOAM to present the meshing result.

Next, while open-source software is rising with its own benefit but it is also having a lot of disadvantage, for example, this software is not easy to use by somebody that didn't masterly in C++ and about programming. This is because most of the open-source software is handling with coding. So it is difficult to know and understand exactly how to handle the procedure in using this software without deep studies.

The other software is commercial package software that provide with paid license and the user need to pay to use every version of it. Every commercial package software was coming with solution manual and tutorial as graphically to make the user easily understand how to use it. This commercial software has icon for each function and it is easily to use as the data and parameter can be filled directly and didn't need to filled it in programming style.

Besides, the disadvantage of the commercial package software is every version that has been upgraded is coming with expensive price. The user need to spend much more money to buy the upgraded version while want to use the more features of each upgraded version. Therefore, every version has a limited feature that already been provided by the company, if the user need to use more features, the new version can be bought.

5.3 RECOMMENDATIONS

As this research has been conducted and the report are being proposed, some recommendations has been listed in order to make the software or the system more useful in the future. The software is already in good conditions but when some part is being adjusted, the more useful to the user can the software be.

The first recommendations that can be stated is the software designing for the open-source software, it can be added more icon and less the coding style in order to easier the user to use it. Then, provide more features in just one software and didn't need to get more extension software in order to get a feature. It is because most of the already launched open-source software are providing a software with extension-friendly that allow the user to get more features from the other software, so it is make the software user difficult and need to install each software.

Besides, the other recommendations are in future, the founder of open-source software can launch the upgraded open-source software with user manual in order to make the user understand and can get the idea how to use it. Otherwise, the tutorial was included in the website to make the user easily understands about the coding style and know exactly how to make and filled the parameter in the coding.

Next, the recommendations that can be listed is the commercial-package engineering software are usually has been taught in most of the institute in Malaysia by using the student versions, but the open-source software are not being exposed to the student. This proposal that can be proposed to the institute is to use the open-source software in order to make the student proficient in coding programming and to accustom the students with the open-source style. While, the features of commercial-package and open-source software are not too different just different in the way to use it; hopefully after the open-source software are used in worldwide, the institute can produce more engineer programming to upgrade the open-source software to up to the same level with the commercial-package software.

REFFERENCE

Ambrosino, F., & Funel, A. (2006). OpenFOAM and Fluent Features in CFD Simulations on CRESCO High Power Computing System, 6–9.

Center, G. R. (2015). CFD Analysis Process, 1–13.

Collins, L. (n.d.). Introduction to CFD Basics, 1–17.

Conditions, N. B. (2016). The open source CFD toolbox OpenFOAM ® v1606 + : New Boundary Conditions Inflow turbulence generator, 1–5.

Corporation, C. 2017 M. S. (2011). Multibody Dynamics and Kinematics. https://doi.org/10.1007/978-90-481-9971-6

Corporations, D. S. S. (2017). Computational Fluid Dynamics (CFD) Simulation Capabilities in SolidWorks. https://doi.org/21-May-2017

Dynamics, C. F., Dynamics, C. F., Dynamics, C. F., Dynamics, C. F., Dynamics, C. F., & Dynamics, C. F. (n.d.). What is Computational Fluid Dynamics (CFD)? ! Beginning of CFD.

Functionality, N. M. (2016). The open source CFD toolbox OpenFOAM ® v1606 + : New Meshing Functionality snappyHexMesh refinement and unrefinement snappyHexMesh locations in mesh, 12–14.

Haddadi, B., Wien, T. U., Jordan, C., Wien, T. U., Harasek, M., & Wien, T. U. (2015). OpenFOAM ® Basic Training.

Inc., A.-F. (2006). ANSYS-Fluent. In *ANSYS Inc*.

Initiative, O. S. (2013). The Open Source Definition. *Open Source Initiative*, 6–7. Retrieved from http://opensource.org/osd

Kuzmin, I. D. (n.d.). Introduction to Computational Fluid Dynamics.

Lakhani, K. R., & Von Hippel, E. (2003). How open source software works: “free” user-to-user assistance. *Research Policy*, *32*(6), 923–943. https://doi.org/10.1016/S0048-7333(02)00095-1

Lomax, H., Pulliam, T. H., & Zingg, D. W. (1999). Fundamentals of Computational Fluid Dynamics.

Maxwell, A. (n.d.). FLUIDS STRUCTURES ELECTRONICS, 0–9.

Menter, F. R. (2012). Best Practice : Scale - Resolving Simulations in ANSYS CFD.

ANSYS Inc, (April), 1–70. Retrieved from http://ainastran.org/staticassets/ANSYS/staticassets/resourcelibrary/techbrief/tb-best-practices-scale-resolving-models.pdf

Salome OpenFOAM Tutorial - CAD model to Solution Complete. (n.d.).

Swanson, J. A. (2014). ANSYS-Fluent Experiment, 3–4.

Your, R., & Promise, P. (n.d.). Fluid Dynamics.

APPENDICES

APPENDIX A : CFD REPORT IN SOLIDWORKS

Initial Mesh Settings

Automatic initial mesh: On

Result resolution level: 5

Advanced narrow channel refinement: Off

Refinement in solid region: Off

Geometry Resolution

Evaluation of minimum gap size: Automatic

Evaluation of minimum wall thickness: Automatic

Computational Domain

Size

X min	-13.592 m
X max	47.692 m
Y min	-14.112 m
Y max	15.989 m
Z min	0 m
Z max	21.065 m

Boundary Conditions

2D plane flow	None
At X min	Default
At X max	Default
At Y min	Default
At Y max	Default
At Z min	Default
At Z max	Default

Physical Features

Heat conduction in solids: Off

Time dependent: Off

Gravitational effects: Off

Flow type: Laminar and turbulent

High Mach number flow: Off

Humidity: Off

Default roughness: 0 micrometer

Default wall conditions: Adiabatic wall

Ambient Conditions

Thermodynamic parameters	Static Pressure: 101325.00 Pa Temperature: 293.20 K
Velocity parameters	Velocity vector Velocity in X direction: 133.000 m/s Velocity in Y direction: 0 m/s Velocity in Z direction: 0 m/s
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 % Length: 0.043 m

Material Settings

Fluids

Air

Goals

Global Goals

GG Av Static Pressure 1

Type	Global Goal
Goal type	Static Pressure
Calculate	Average value
Coordinate system	Global coordinate system
Use in convergence	On

GG Av Total Pressure 1

Type	Global Goal
Goal type	Total Pressure
Calculate	Average value
Coordinate system	Global coordinate system
Use in convergence	On

GG Av Dynamic Pressure 1

Type	Global Goal
Goal type	Dynamic Pressure
Calculate	Average value
Coordinate system	Global coordinate system
Use in convergence	On

GG Av Temperature (Fluid) 1

Type	Global Goal
Goal type	Temperature (Fluid)
Calculate	Average value
Coordinate system	Global coordinate system
Use in convergence	On

GG Av Density 1

Type	Global Goal
Goal type	Density
Calculate	Average value
Coordinate system	Global coordinate system
Use in convergence	On

GG Force (X) 1

Type	Global Goal
Goal type	Force (X)
Coordinate system	Global coordinate system
Use in convergence	On

Calculation Control Options

Finish Conditions

Finish conditions	If one is satisfied
Maximum iterations	20
Maximum travels	4
Goals convergence	Analysis interval: 5e-001

Solver Refinement

Refinement: Disabled

Results Saving

Save before refinement	On

Advanced Control Options

Flow Freezing

Flow freezing strategy	Disabled

APPENDIX B: CFD REPORT IN ANSYS

Fluent
Version: 3d, pbns, ske (3d, pressure-based, standard k-epsilon)
Release: 15.0.7
Title:

Boundary Conditions

Zones

name	id	type
wall-surrounding_flow_channel	1	wall
inlet_velocity	6	velocity-inlet
outlet_pressure	7	pressure-outlet

Setup Conditions

wall-surrounding_flow_channel

Condition	Value
Enable shell conduction?	no
Wall Motion	0
Shear Boundary Condition	0
Define wall motion relative to adjacent cell zone?	yes
Apply a rotational velocity to this wall?	no
Velocity Magnitude (m/s)	0
X-Component of Wall Translation	1
Y-Component of Wall Translation	0
Z-Component of Wall Translation	0
Define wall velocity components?	no
X-Component of Wall Translation (m/s)	0
Y-Component of Wall Translation (m/s)	0
Z-Component of Wall Translation (m/s)	0
Wall Roughness Height (m)	0
Wall Roughness Constant	0.5
Rotation Speed (rad/s)	0
X-Position of Rotation-Axis Origin (m)	0
Y-Position of Rotation-Axis Origin (m)	0
Z-Position of Rotation-Axis Origin (m)	0
X-Component of Rotation-Axis Direction	0
Y-Component of Rotation-Axis Direction	0
Z-Component of Rotation-Axis Direction	1
X-component of shear stress (pascal)	0
Y-component of shear stress (pascal)	0
Z-component of shear stress (pascal)	0
Fslip constant	0
Eslip constant	0
Specularity Coefficient	0
Enable Thermal Stabilization?	no
Scale Factor	0
Stabilization Method	1

inlet_velocity

```
Condition                                       Value
--------------------------------------------------
Velocity Specification Method                   2
Reference Frame                                 0
Velocity Magnitude (m/s)                        133
Supersonic/Initial Gauge Pressure (pascal)      0
Coordinate System                               0
X-Velocity (m/s)                                0
Y-Velocity (m/s)                                0
Z-Velocity (m/s)                                0
X-Component of Flow Direction                   1
Y-Component of Flow Direction                   0
Z-Component of Flow Direction                   0
X-Component of Axis Direction                   1
Y-Component of Axis Direction                   0
Z-Component of Axis Direction                   0
X-Coordinate of Axis Origin (m)                 0
Y-Coordinate of Axis Origin (m)                 0
Z-Coordinate of Axis Origin (m)                 0
Angular velocity (rad/s)                        0
Turbulent Specification Method                  2
Turbulent Kinetic Energy (m2/s2)                1
Turbulent Dissipation Rate (m2/s3)              1
Turbulent Intensity (%)                         5
Turbulent Length Scale (m)                      1
Hydraulic Diameter (m)                          1
Turbulent Viscosity Ratio                       10
is zone used in mixing-plane model?             no
```

```
outlet_pressure

   Condition                                              Value
   -------------------------------------------------------------
   Gauge Pressure (pascal)                                0
   Backflow Direction Specification Method                1
   Coordinate System                                      0
   X-Component of Flow Direction                          1
   Y-Component of Flow Direction                          0
   Z-Component of Flow Direction                          0
   X-Component of Axis Direction                          1
   Y-Component of Axis Direction                          0
   Z-Component of Axis Direction                          0
   X-Coordinate of Axis Origin (m)                        0
   Y-Coordinate of Axis Origin (m)                        0
   Z-Coordinate of Axis Origin (m)                        0
   Turbulent Specification Method                         2
   Backflow Turbulent Kinetic Energy (m2/s2)              1
   Backflow Turbulent Dissipation Rate (m2/s3)            1
   Backflow Turbulent Intensity (%)                       5
   Backflow Turbulent Length Scale (m)                    1
   Backflow Hydraulic Diameter (m)                        1
   Backflow Turbulent Viscosity Ratio                     10
   is zone used in mixing-plane model?                    no
   Radial Equilibrium Pressure Distribution               no
   Average Pressure Specification?                        no
                                                          0
   Specify targeted mass flow rate                        no
```

```
        Targeted mass flow (kg/s)                          1
        Upper Limit of Absolute Pressure Value (pascal)    5000000
        Lower Limit of Absolute Pressure Value (pascal)    1

Fluent
Version: 3d, pbns, ske (3d, pressure-based, standard k-epsilon)
Release: 15.0.7
Title:

Cell Zone Conditions
--------------------

   Zones

      name                        id   type
      -------------------------------------
      surrounding_flow_channel    3    fluid

   Setup Conditions

      surrounding_flow_channel

         Condition
Value
         ------------------------------------------------------------
---
         Material Name
air
         Specify source terms?                                        no
         Source Terms                                                 ()
         Specify fixed values?                                        no
         Local Coordinate System for Fixed Velocities                 no
         Fixed Values                                                 ()
         Frame Motion?                                                no
         Relative To Cell Zone                                        -1
         Reference Frame Rotation Speed (rad/s)                       0
         Reference Frame X-Velocity Of Zone (m/s)                     0
         Reference Frame Y-Velocity Of Zone (m/s)                     0
         Reference Frame Z-Velocity Of Zone (m/s)                     0
         Reference Frame X-Origin of Rotation-Axis (m)                0
         Reference Frame Y-Origin of Rotation-Axis (m)                0
         Reference Frame Z-Origin of Rotation-Axis (m)                0
         Reference Frame X-Component of Rotation-Axis                 0
         Reference Frame Y-Component of Rotation-Axis                 0
         Reference Frame Z-Component of Rotation-Axis                 1
         Reference Frame User Defined Zone Motion Function
none
         Mesh Motion?                                                 no
         Relative To Cell Zone                                        -1
         Moving Mesh Rotation Speed (rad/s)                           0
         Moving Mesh X-Velocity Of Zone (m/s)                         0
         Moving Mesh Y-Velocity Of Zone (m/s)                         0
         Moving Mesh Z-Velocity Of Zone (m/s)                         0
         Moving Mesh X-Origin of Rotation-Axis (m)                    0
         Moving Mesh Y-Origin of Rotation-Axis (m)                    0
         Moving Mesh Z-Origin of Rotation-Axis (m)                    0
         Moving Mesh X-Component of Rotation-Axis                     0
         Moving Mesh Y-Component of Rotation-Axis                     0
```

```
          Moving Mesh Z-Component of Rotation-Axis                  1
          Moving Mesh User Defined Zone Motion Function
none
          Deactivated Thread                                        no
          Laminar zone?                                             no
          Set Turbulent Viscosity to zero within laminar zone?
yes
          Embedded Subgrid-Scale Model                              0
          Momentum Spatial Discretization                           0
          Cwale
0.325
          Cs
0.1
          Porous zone?                                              no
          Conical porous zone?                                      no
          X-Component of Direction-1 Vector                         1
          Y-Component of Direction-1 Vector                         0
          Z-Component of Direction-1 Vector                         0
          X-Component of Direction-2 Vector                         0
          Y-Component of Direction-2 Vector                         1
          Z-Component of Direction-2 Vector                         0
          X-Component of Cone Axis Vector                           1
          Y-Component of Cone Axis Vector                           0
          Z-Component of Cone Axis Vector                           0
          X-Coordinate of Point on Cone Axis (m)                    1
          Y-Coordinate of Point on Cone Axis (m)                    0
          Z-Coordinate of Point on Cone Axis (m)                    0
          Half Angle of Cone Relative to its Axis (deg)             0
          Relative Velocity Resistance Formulation?
yes
          Direction-1 Viscous Resistance (1/m2)                     0
          Direction-2 Viscous Resistance (1/m2)                     0
          Direction-3 Viscous Resistance (1/m2)                     0
          Choose alternative formulation for inertial resistance?   no
          Direction-1 Inertial Resistance (1/m)                     0
          Direction-2 Inertial Resistance (1/m)                     0
          Direction-3 Inertial Resistance (1/m)                     0
          C0 Coefficient for Power-Law                              0
          C1 Coefficient for Power-Law                              0
          Porosity                                                  1
          Interfacial Area Density (1/m)                            1
          Heat Transfer Coefficient (w/m2-k)                        1
          3D Fan Zone?                                              no
          Inlet Fan Zone                                            0
          Fan Thickness (m)                                         0
          Fan Inner Diameter (m)                                    0
          Fan Outer Diameter (m)                                    0
          X-Component of 3D Fan Origin (m)                          0
          Y-Component of 3D Fan Origin (m)                          0
          Z-Component of 3D Fan Origin (m)                          0
          Rotational Direction                                      0
          Fan Operating Angular Velocity (rad/s)                    0
          Fan Inflection Point                                      0
          Limit Flow Rate Through Fan                               no
          Maximum Flow Rate (m3/s)                                  0
          Minimum Flow Rate (m3/s)                                  0
          Tangential Source Term                                    no
          Radial Source Term                                        no
```

```
          Axial Source Term                                           no
          Method                                                      0
          Pressure Jump (pascal)                                      0
          Fan Curve Fitting Method                                    0
          Polynomial Order                                            0
          Initial Flow Rate (m3/s)                                    0
          Fan Test Angular Velocity (rad/s)                           0
          Fan Test Temperature (k)                                    0
          Read Fan Curve                                              no


Fluent
Version: 3d, pbns, ske (3d, pressure-based, standard k-epsilon)
Release: 15.0.7
Title:

Solver Settings
---------------

   Equations

      Equation     Solved
      -------------------
      Flow         yes
      Turbulence   yes

   Numerics

      Numeric                         Enabled
      ---------------------------------------
      Absolute Velocity Formulation   yes

   Relaxation

      Variable                     Relaxation Factor
      ----------------------------------------------
      Pressure                     0.3
      Density                      1
      Body Forces                  1
      Momentum                     0.7
      Turbulent Kinetic Energy     0.8
      Turbulent Dissipation Rate   0.8
      Turbulent Viscosity          1

   Linear Solver

                                   Solver     Termination    Residual
Reduction
      Variable                     Type       Criterion      Tolerance
      ----------------------------------------------------------------
---------
      Pressure                     V-Cycle    0.1
      X-Momentum                   Flexible   0.1            0.7
      Y-Momentum                   Flexible   0.1            0.7
      Z-Momentum                   Flexible   0.1            0.7
      Turbulent Kinetic Energy     Flexible   0.1            0.7
      Turbulent Dissipation Rate   Flexible   0.1            0.7
```

```
   Pressure-Velocity Coupling

      Parameter   Value
      ------------------
      Type        SIMPLE

   Discretization Scheme

      Variable                      Scheme
      ------------------------------------------------
      Pressure                      Second Order
      Momentum                      Second Order Upwind
      Turbulent Kinetic Energy      Second Order Upwind
      Turbulent Dissipation Rate    First Order Upwind

   Solution Limits

      Quantity                          Limit
      ---------------------------------------
      Minimum Absolute Pressure         1
      Maximum Absolute Pressure         5e+10
      Minimum Temperature               1
      Maximum Temperature               5000
      Minimum Turb. Kinetic Energy      1e-14
      Minimum Turb. Dissipation Rate    1e-20
      Maximum Turb. Viscosity Ratio     100000

                                         "Force Report"

Center of Pressure - Set Coordinate x = 0 (m)
Zone                         y                 z
wall-surrounding_flow_channel -7.9239118      10.081884
-------------------------- --------------- ---------------
Net                        -7.9239118      10.081884
```