8-2011

CFD ANALYSIS OF THE UNDER HOOD OF A CAR FOR PACKAGING CONSIDERATIONS

Sreekanth reddy Gondipalle
Clemson University, sgondip@clemson.edu

CFD ANALYSIS OF THE UNDER HOOD OF A CAR FOR PACKAGING CONSIDERATIONS

A Thesis
Presented to
the Graduate School of
Clemson University

In Partial Fulfillment
of the Requirements for the Degree
Master of Science
Mechanical Engineering

by
Sreekanth Reddy Gondipalle
August 2011

Accepted by:
Dr. Georges M. Fadel, Committee Chair
Dr. Richard S. Miller
Dr. Rui Qiao

ABSTRACT

In recent years, there has been an increase in demand towards the improvement of car design for achieving better performance and increasing passenger comfort. Improving the design of individual components to meet the customer needs for improved vehicle performance alone is not enough. Interactions of these components with the surrounding components and their placement should also be investigated. Placement of these components in the under hood space forms a 3-Dimensional packaging problem. In the past, a multi objective optimization process was setup to determine the optimal placement of these components in the car under hood space. Three main objectives were taken into account namely, minimizing center of gravity height, maximizing vehicle maintainability and maximizing survivability in the optimization process. However, minimizing the overall under hood temperature and ensuring the temperature of heat sensitive components to be below its critical value, is not added as an objective to the optimization problem.

This study makes an assessment of the need for including the thermal objective into the optimization process and also presents an efficient way of performing CFD simulation over the under hood geometry. The under hood geometry used included radiator, engine, exhaust manifold, coolant tank, air filter, brake booster, front grille geometry and battery. These components were included as heat source, heat exchangers etc. A standard k-ε

turbulence model with upward differencing convection scheme is used on a well refined computational mesh.

The work also describes in detail the way of accurately and effectively modeling the radiator as an ungrouped macro heat exchanger model available in Ansys FLUENT. The results obtained from the CFD simulations illustrate the importance of the under hood vehicle configuration optimization process on its thermal behavior. The temperature attained by the coolant flowing through the radiator with constant heat rejection, when placed behind the engine is very high, when compared to the temperature it attained with the radiator placed in front of the engine. The CFD analysis presented in this study is performed using Ansys FLUENT while the initial geometry preparation is done using SolidWorks.

The CFD analysis presented in this work is then used to build an approximation by my research mate, which is later tied to an optimizer based on Genetic Algorithm. Thus, including the thermal objective to the multi objective optimization problem stated above.

DEDICATION

To my parents, sister and all my well-wishers for all the love and support they had given to me throughout my life.

ACKNOWLEDGMENTS

I would like to express my sincere gratitude and thanks to Dr. Georges M. Fadel for his valuable guidance and support through my research efforts. I would also like to thank Dr. Richard S. Miller and Dr. Rui Qiao for serving on my research advisory committee.

I would like to thank Dr. Paolo Guarneri for extending his support, encouragement and providing valuable inputs in completing this project. I would also like to thank Raviteja Katragadda, my fellow research mate, for his help and encouragement throughout my research work.

I would specially like to thank the Automotive Research Center for sponsoring this research. Finally, I would like to thank all my colleagues and friends in the Mechanical Engineering Department at Clemson University for their help and support when it was needed.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER ONE

INTRODUCTION

In recent years, there has been an increase in demand towards the improvement of car design for achieving better performance and increasing passenger comfort. A large effort was invested in improving the aerodynamics of the car to attain reduced drag with improved acceleration and fuel economy. Advancements were made in the suspension design to improve maneuverability. Also, the engine design was optimized to get better fuel efficiencies and greater power. Thus, a large effort was made to improve the design of individual components to meet the demands for improved performance.

Now, placing components in the under hood space of a car makes the space crowded. In such crowded environments, performance of one individual component may affect the performance of several others in its vicinity. Thus, enhancing the design of individual components alone is not enough to improve the vehicle performance but also their placements under the hood of a car and their interactions with the components surrounding them have to be investigated. An optimization process should be applied to this packaging problem to find an optimal placement of these components. This under hood vehicle configuration design problem is a complex multi objective optimization problem. In the past, this multi objective optimization problem with three major objectives namely, minimizing center of gravity height, maximizing vehicle maintainability and maximizing survivability was investigated [1].

However, research was not done on including the thermal aspects of the car under hood as an objective for this complex optimization problem. With the evolution of better car designs to cater to the needs of the customer, the under hood space of an automobile car was confined to a much smaller and compact space. But, several heat generating components such as engine, exhaust manifold etc., placed in this confined under hood space can adversely affect the overall temperature of the under hood region of a car. Thus, minimizing the overall temperature of the car under hood should be added as an objective to the multi objective optimization problem discussed above.

The under hood thermal management might look simple, but with the engine size being increased for higher power outputs, the under hood space being confined to smaller volumes for compactness and also with more and more components being clustered in to the under hood, there is not enough space for the air to flow through it. With this extremely complicated and tightly packed under hood environment, it is not easy to carry out experimental studies to predict the temperatures obtained in this region. Also to assist early design changes, it is not always plausible to carry out these experimental simulations by building prototypes for different designs, as it would increase the design cycle time. Thus, we need to carry out numerical simulations to understand and improve the thermal behavior of the car under hood.

Most designers resort to the technology known as CFD (computational fluid dynamics) to carry out these numerical simulations. This is a powerful tool to model fluid flow and understand the thermal behavior numerically using the finite volume methods. This tool can also be used to describe and simulate complex thermal fluid flow phenomena such as turbulent flow, flow through porous medium, and flow past heat exchangers. But, integrating CFD simulations with the optimization process is computationally expensive. Thus, an approximation to the CFD simulation should be built for integrating the thermal model with the optimization process.

In this project, we investigate the airflow and thermal behavior in the under hood region of a car by carrying out CFD simulations. Thus, our goal for this project is to come up with a CFD simulation of the complex 3-Dimensional under hood space, whose results can later be used to build a neural network approximation to it.

CHAPTER TWO

LITERATURE REVIEW

2.1 CFD (Computational Fluid Dynamics)

CFD is a numerical simulation tool to predict the air flow characteristics and its associated thermal phenomena in a 3-Dimensional or a 2-Dimensional space. The method it follows to get the results is that it discretizes the domain space into several small volumes over which the fundamental equations are solved for the velocity and temperature profiles. The equations that are solved over the domain are the energy equation, momentum equation (Navier stokes equation) and the continuity equation. CFD uses finite volume discretization techniques for dividing the entire domain space into several small volumes. Discretization of the space into smaller volumes is called mesh generation. The size of the cells into which the domain is discretized depends on the users need for accuracy of the solution in that domain.

For our problem, as mentioned earlier, we need to perform CFD simulation over the car under hood space. For this we use the commercial CFD simulation tool FLUENT. FLUENT is the fluid flow simulation tool of ANSYS multiphysics software. In the past there has been an extensive work done on simulating the thermal behavior of the car under hood using various simulation tools.

2.2 RELEVANT WORK

Salvio Chacko et al. [2] presented a methodology for performing a CFD analysis to predict the air flow and temperature distribution through the vehicle under hood. Through the results obtained, suggestions to improve the air flow through the under hood were made. The method presented was illustrated with the work done over the under hood of a light truck. An under hood simulation model which included all the components relevant for the thermal management like radiator, engine, exhaust manifold, condenser, fans, etc. was used. Instead of modeling radiator and grille in detail, which is complicated, they were modeled as a heat exchanger and porous zone respectively. A standard k-ε turbulence model with Upward Differencing convection scheme with radiation effects was used on a properly refined computational grid. Ansys FLUENT was used as a simulation tool while the mesh was generated using TGRID. However, performing the same task over the car under hood is challenging because of its compactness.

Weidmann et al. [3] discuss a method to predict the air flow and temperature distribution in the car under hood. A coupled steady-state CFD run and thermal analysis was conducted using StarCD as the simulation tool to predict the thermal characteristics of the under hood at idle conditions without a working fan. The radiative and conductive heat transfer analyses were conducted using RadTherm. The coupling of the flow field and temperature distribution for flows which have natural convection as the mode of heat transfer requires an iterative process. The iterative process consumed a lot of computational power and time. Later the results obtained were compared to experimental data to validate the simulation method as much as possible.

I.F.Hsu et al. [4] presented a methodology to carry out computational fluid dynamics simulations for understanding the thermal characteristics of the environment surrounding an underbody fuel tank in a passenger vehicle. In this paper, a body-fitted unstructured CFD model of the underbody region which included the fuel tank was used. The results for both moving and stationary cases of the car presented in this paper indicated that the major source of the heat transfer to the fuel tank surface was due to the heat convected from the under hood region of the vehicle. The results were validated with the test data from a similar vehicle.

Wenyan et al. [5] discuss a methodology to carry out the under hood cooling simulation of a lift truck. A brief description of the challenges faced in designing that cooling system is presented. The tight space in the engine compartment was identified as the major cause for overheating issues in the under hood region. CFD simulations were identified as an advantageous tool to visualize flow fields and verify design at initial stages. To perform these simulations, the CAD geometry was obtained using ProE, ANSA was used for geometry cleanup and surface mesh generation, Tgrid for generating tetra volume mesh and FLUENT to do the simulations. In the simulations performed, the under hood region was discretized into 1 million cells. The radiator was modeled as a heat exchanger and the fan as a moving reference frame model in FLUENT. From the simulations coolant water temperatures were predicted and the plots of flow and thermal fields on a plane cut across the under hood region were generated.

Srinivasan et al. [6] present an efficient procedure for the vehicle thermal protection development process. They recognize the significance of the use of CFD simulations during the design phases for early detection of thermal issues even before building the vehicle prototypes for experimental evaluations. In the current work, an interior-to-boundary method mesh generation technique was used unlike the traditional mesh generation techniques which were time consuming. The method proposed was illustrated with the work done on a passenger vehicle underbody model. Results were validated with test data.

Bancroft et al. [7] present the importance of the initial geometry preparation by computer Aided Engineering (CAE) tools before running CFD simulations over it. It was shown that there were significant reductions in time taken to generate a CFD mesh after the CAD geometry cleanup operations were performed. It was mentioned that, however good the quality of the CAD geometry may be, it still needs to be processed before performing CFD simulations over it. Later the CFD simulations were performed on the processed CAD data and were presented in this paper. Validations for the CFD simulations with test data were also presented which showed a good correlation between them.

Huang et al. [8] had investigated the sensitivity of the size of a flow domain to the air flow rates through the radiator. From the results obtained, a general method for choosing the optimal size of a flow domain to perform under hood air flow simulations using CFD codes was proposed. Finally, the air flow rates simulated by using the proposed optimal flow domain size was compared with the experimentally determined values. Both these data showed a good correlation between them.

Vinod et al. [9] studied the significance of the effect of air flow through the radiator. Radiator efficiency was improved by optimizing the air flow through it. The improvement in efficiency was visualized through various numerical simulations that were performed. These CFD simulations were conducted using FLUENT, while the surface and the volume mesh were generated using ANSA and Tgrid, respectively. Later the results were validated with the test data.

Winnard et al. [10] presented a series of tests that were conducted by varying different parameters that affect the under hood thermal conditions. The parameters that were varied are

- Diverting the fan airflow from the engine compartment
- Forced cooling of the exhaust manifold which is the major heat source available in the vehicle under hood region.

These tests were conducted on a Ford F-250 Light Truck. The results indicated that there was a decrease in the under hood average temperatures by diverting the outlet air from the radiator and also by forced cooling of the exhaust manifold. These results thus indicate that there is a significant improvement of the thermal behavior with the changes made in the design.

2.3 DISCUSSION

The literature presented describes the challenges that the researchers faced in the past in performing CFD simulations over the complex 3-Dimensional vehicle under hood geometries. Also suggestions to overcome these challenges were made in the literature.

The general procedure to follow to perform these rigorous CFD simulations over the complex under hood geometry as inferred from the literature is summarized as follows. Initially the CAD geometry of good quality has to be imported into the CFD simulation tools in an acceptable format. The imported geometry should then be processed by performing the geometry cleanup operations to make it usable for CFD simulations. The processed geometry is then meshed with the desired grid type.

Form the results presented in Winnard et al. [10] it can be inferred that the amount of heat transferred by radiation from exhaust manifold constitutes only a fraction of the total amount of heat transferred to the under hood environment. This inference holds true if it is assumed that the results from this paper are applicable to all vehicles.

As presented in Salvio Chacko et al. [2], the radiator can be modeled as a porous model in FLUENT to further simplify and reduce the time taken to perform the CFD simulations without any sacrifice being made in the accuracy of the solutions. The size of the flow domain on which the CFD simulations are made can be chosen according to the method presented in Huang et al. [8].

Finally from the literature study done, it can be inferred that in the past, researchers aimed at enhancing the automobile manufacturing process by performing early design evaluations with the use of computer aided engineering tools such as CFD and FEM. This reduced the costly prototyping process for evaluating each new design and significantly impacted the design cycle time.

CHAPTER THREE

PROBLEM SETUP

3.1 OVERVIEW

This chapter describes the procedure followed in setting up the problem for performing CFD simulations. In brief, the sequence of steps followed in solving the problem is presented in a flowchart as shown in the Figure 3.1

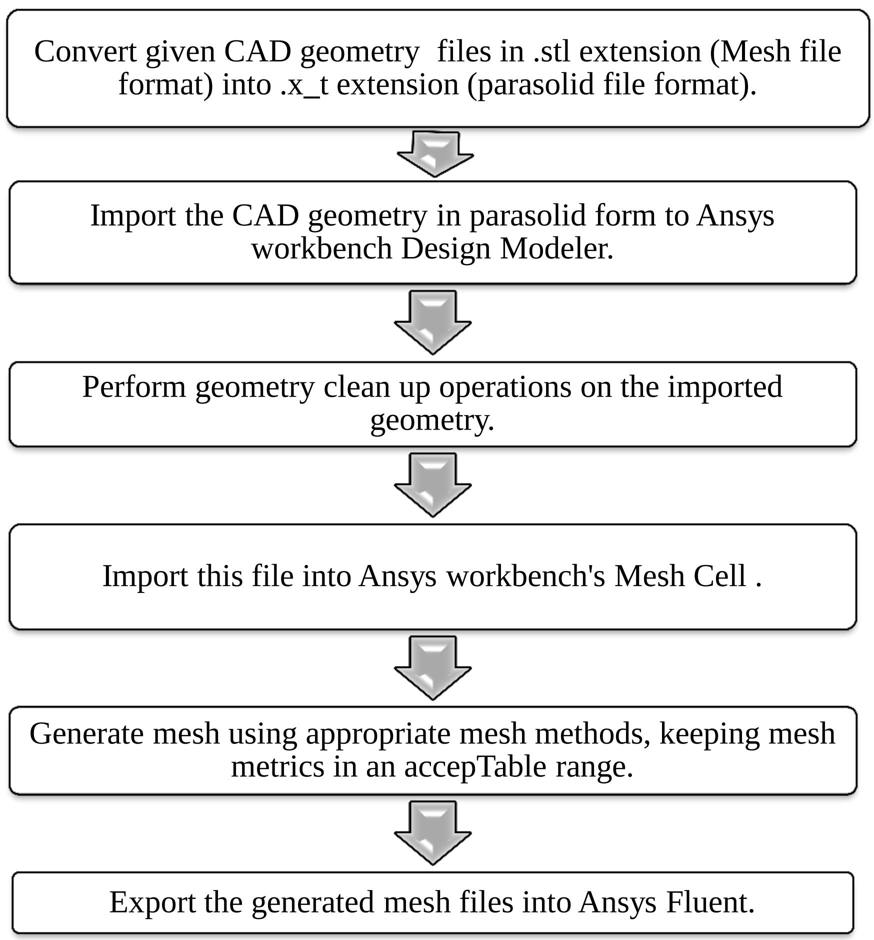


Figure 3.1: Flow chart presenting the steps followed in setting up the problem for CFD simulation.

3.2 INITIAL GEOMETRY PREPARATION

The CAD geometries of the under hood components of a car were provided in the tessellated file (.stl extension) format, for use with this project. This is a common format that can be generated by all solid modelers. However, Ansys FLUENT design modeler cannot import .stl type extension files. Instead, it can import the following external geometry file models [11]

- Parasolid (extension .x_t & .xmt_txt or .x_b & .xmt_bin)
- IGES (extension .igs or .iges)
- Monte Carlo N-Particle (extension .mcnp)
- ACIS (extension .sat)
- BladeGen (extension .bgd)
- .STEP (extension .step and .stp)

Thus, the geometry in tessellated file format was converted into the Parasolid format, using SolidWorks software's Scanto3D functionality. Scanto3D tool in SolidWorks software significantly reduces the time taken to build complex 3D models from the mesh file [12]. Conversion of the mesh file into a 3D geometric model is a three step process as shown in Figure 3.2 [12].

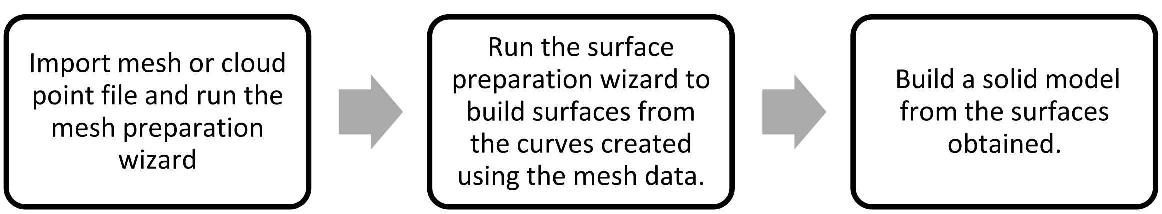
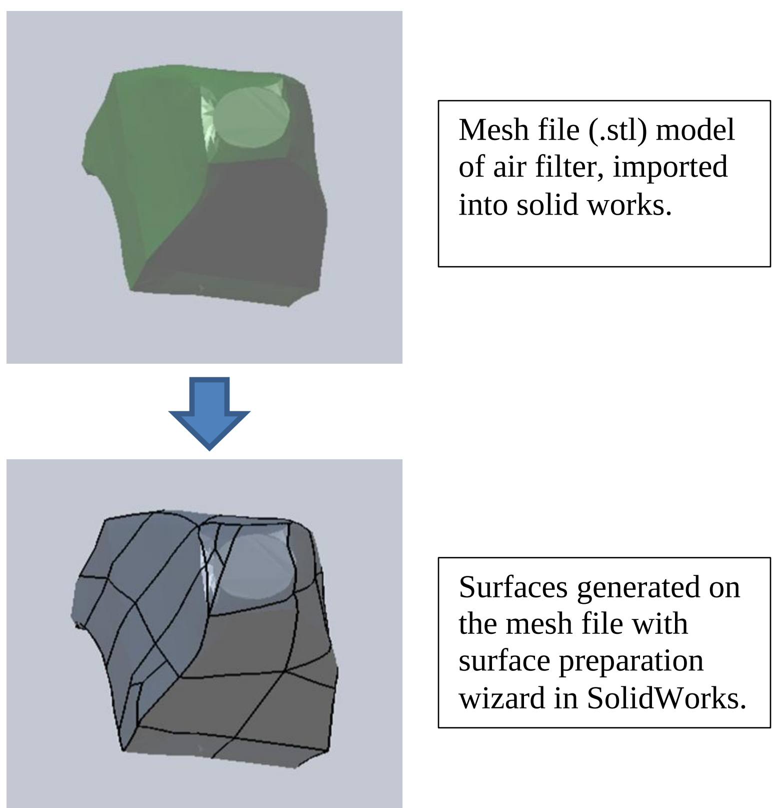


Figure 3.2: Sequence in which the Solid Model is built using Scanto3D tool.

Once the mesh file is imported, using the Scanto3D tool in SolidWorks software, the mesh preparation wizard is run. The mesh preparation wizard prompts the user to [12]

- Specify the alignment of the geometry.
- Reduce the element noise to desired level.
- Simplify the geometry by smoothening till desired levels.
- Fill holes if necessary.

Once, the mesh preparation wizard is completed, then the surface preparation wizard is run. It prompts the user to specify the method it has to follow to build the surfaces and resulting solid model. Automatic solid creation method was chosen to convert the mesh data of all the under hood components into solid models. The conversion of the mesh file of an air filter component of the car into solid model can be seen pictorially in Figure 3.3. The converted solid model is then saved as a Parasolid file.



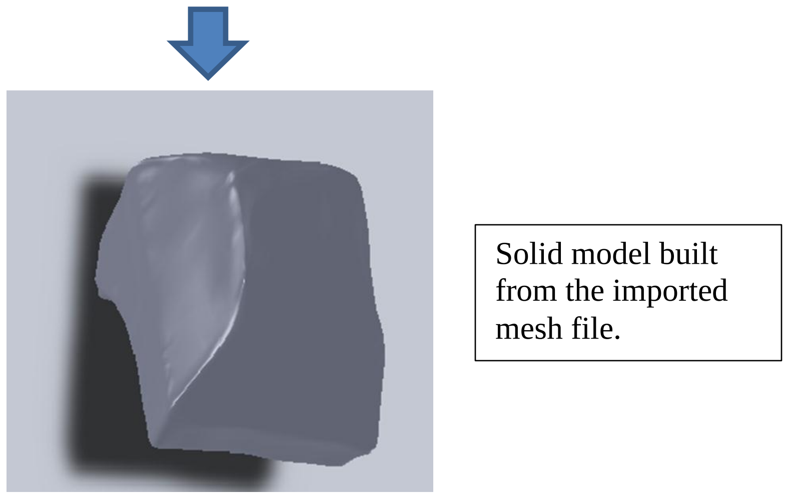


Figure 3.3: Summary of the initial geometry preparation

3.3 GEOMETRY CLEANUP PROCESS

Thus, as mentioned above, the geometry was imported into the Ansys Design Modeler as a Parasolid file. The imported geometry is then checked for problematic geometry within it. Clean up and repair of this problematic geometry within the CAD data is a vital step in setting up the problem for CFD simulations.

Ansys Design Modeler has a set of semi-automatic tools to search for problematic geometrical features and repair them [13]. These tools search for the following eight types of problematic geometric features

- Hard Edges: The edges that do not form the boundary of the face are called hard edges. These edges are undesirable, as they cause an unwanted concentration of mesh near them.

- Short Edges: Edges that are too short have to be removed from the model. An edge whose length is below a certain minimum limit is determined to be a faulty geometric feature. This limit can be decided by the user if more accuracy is desired.
- Seams: Ansys Design Modeler generally defines seams as a gap formed between connected laminar edges [13]. These small gaps produce undesirable effects while meshing and thus need to be eliminated. The maximum width of the gap between the edges is used as a criterion for search. If this width lies below a minimum limit determined by the Design Modeler, then it is determined to be a faulty geometry.
- Holes: If the holes present on either surface bodies or solid bodies are not of much importance in the analysis of the entire system, considering their size in comparison with the entire system's size, these holes can be eliminated by filling up the cavities, to reduce the complexity involved in building the mesh.
- Sharp angles: The presence of very small angles on a face, which need not be studied in detail, should be eliminated. They cause unnecessary complexity in the meshing process.
- Slivers: The Ansys Design Modeler defines a sliver as a very narrow face with two or more edges [13]. The maximum width of the face is used as the criteria to search for the sliver faces. If this maximum width is less than a certain limit set by the user according to his desired accuracy, then, the face is merged with one of its large adjacent faces.

- Spikes: spikes are defined as the narrow areas of a large face which might create complexities in meshing. These spikes are eliminated by merging the face with an adjacent face.
- Faces: Faces with very small area can also lead to an undesirable level of concentration of the mesh in that zone. If this level of accuracy is not desired in meshing, then, the faces with small areas are merged with their adjacent faces.

As mentioned above, for the geometric features to be determined as faulty features, their search criterion needs to be in the range defined by the user. Thus, the user needs to perform a tradeoff between desired accuracy and reduction of time taken to mesh the part and specify the limits in which the search criterion, corresponding to a feature, has to fall in order to be determined as problematic geometry.

In this project, the under hood geometry of the car was checked for the problematic geometric features using the semi-automatic tools available in Ansys Design Modeler. These semi-automatic tools recognized that, in the geometry provided, two parts named “Underhood-shell” and “Airfilter”, had problematic geometric features associated with them. Upon visual inspection, it was determined that the problematic features associated with the part “Underhood-shell” need not be repaired, as its presence would not create any hassle while meshing. But, the faulty geometric feature i.e. slivers, associated with the part “Airfilter” was repaired using an automatic method provided by the Ansys Design Modeler. This cleanup process significantly reduced the time taken to mesh this part. Figure 3.4 shows the sliver present on the CAD geometry of Airfilter.

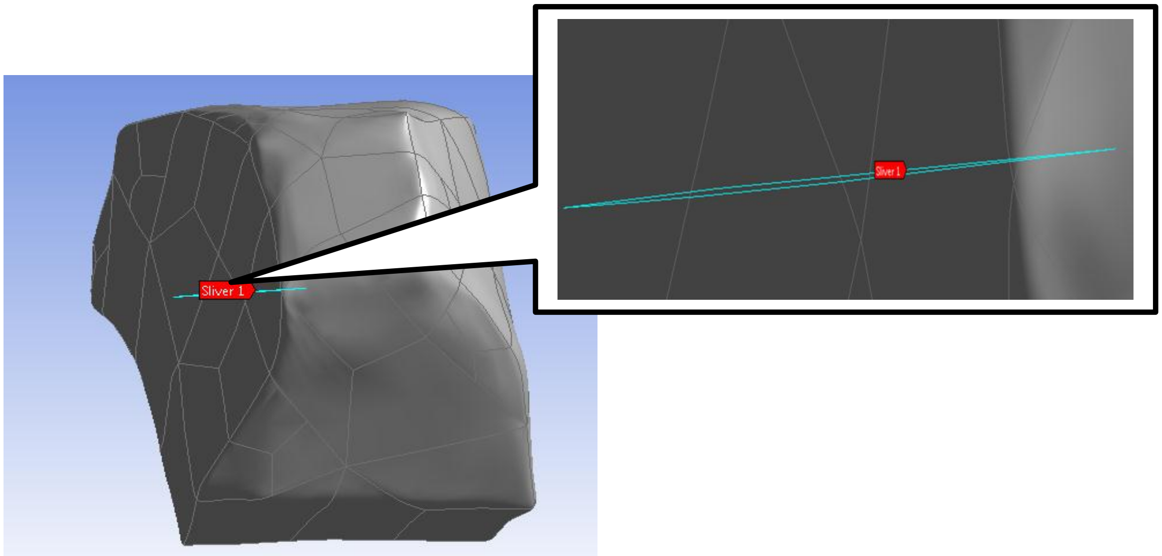


Figure 3.4: CAD geometry of Airfilter, before the sliver was eliminated

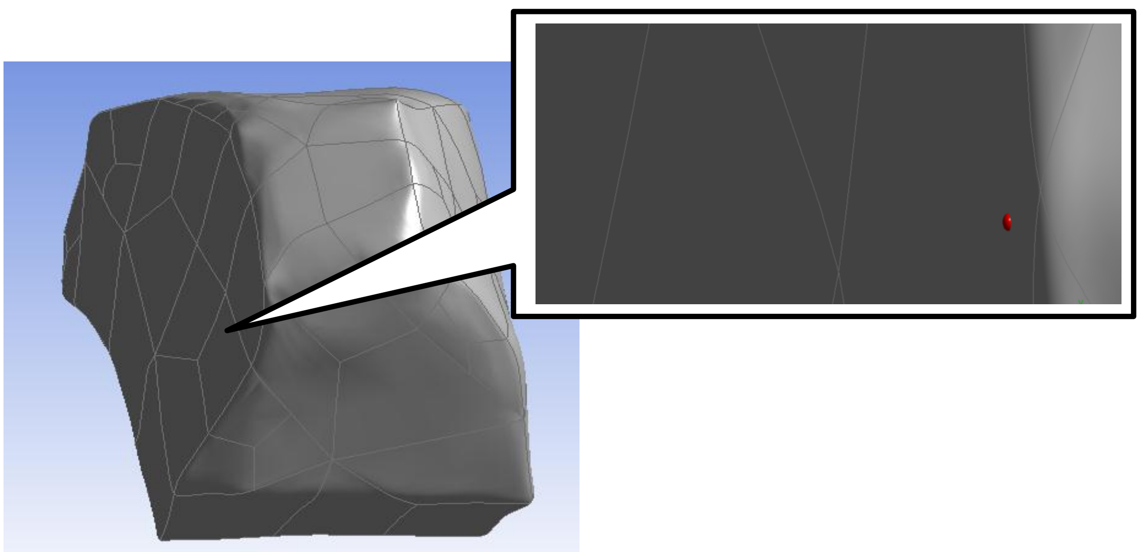


Figure 3.5: CAD geometry of Airfilter, after the sliver was eliminated

Thus, the 3-dimensional CAD geometry of the car under hood used for performing CFD simulations in this project is shown below in Figures 3.6 to 3.8. In general, the under hood compartment of a car contains a complex web of piping and wiring system occupying the empty spaces between the major components. But, to perform CFD simulations, building the CAD geometry of this complex piping and wiring system can be neglected, assuming their effects to be negligible on the under hood thermal behavior [2].

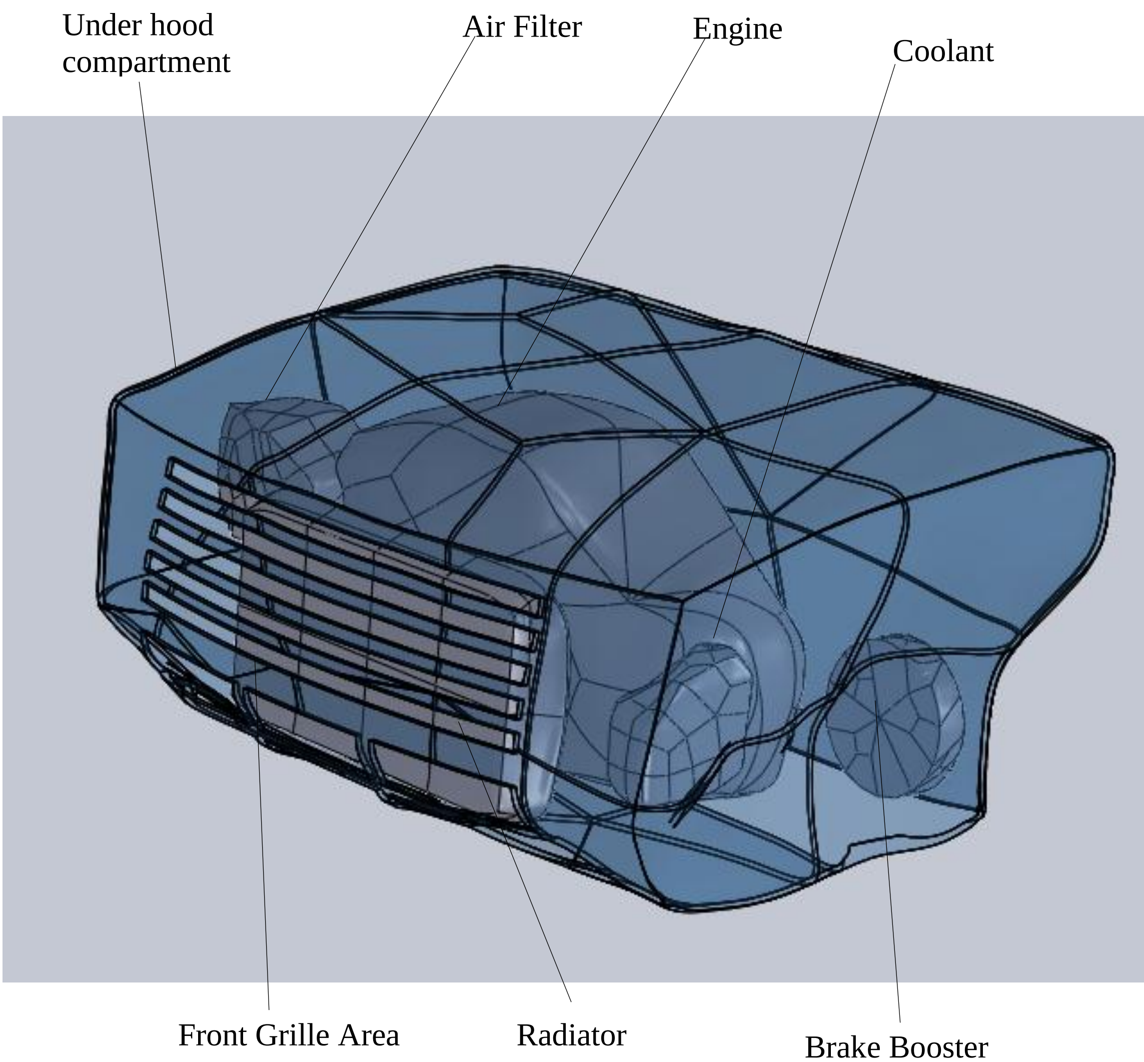


Figure 3.6: Perspective view of a 3-Dimensional CAD model of the car under hood, used for CFD simulations.

The geometric model used to perform CFD simulations included the air filter, engine, radiator, exhaust manifold, battery, brake booster and under hood compartment. Figure 3.7 displays the under hood geometry with the grille removed in front view and Figure 3.8 displays it in top view.

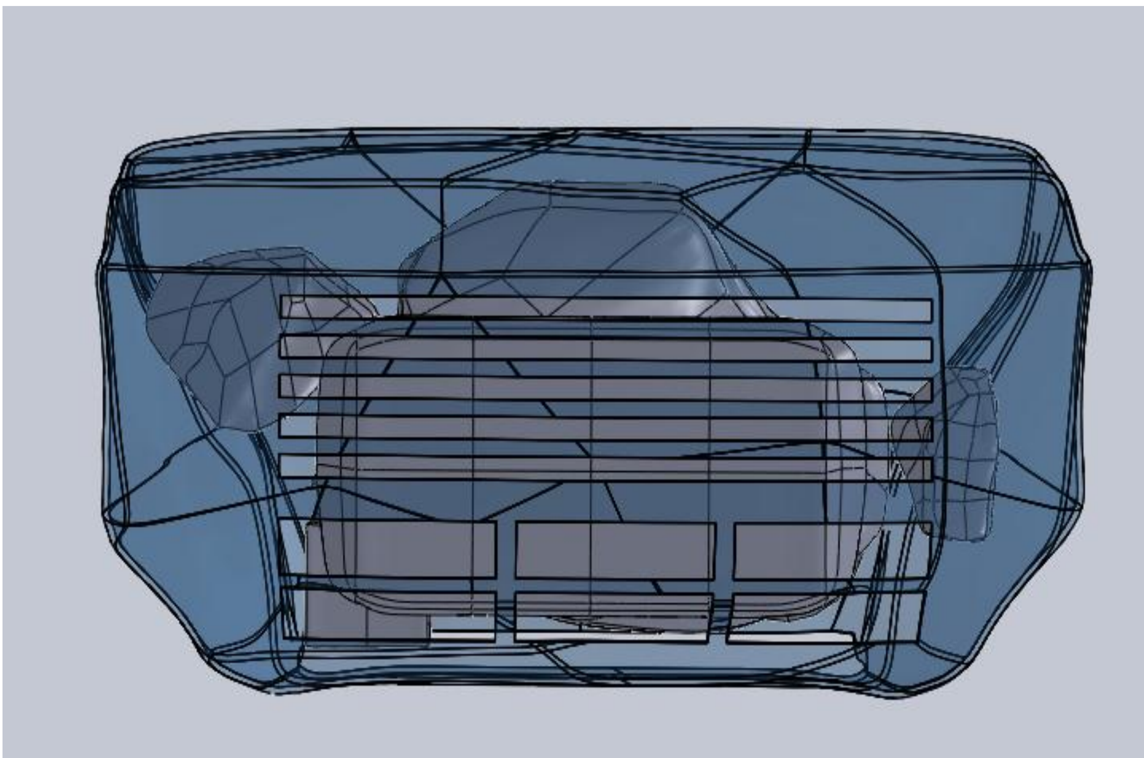


Figure 3.7: Front view of a 3-Dimensional CAD model of the car under hood, used for CFD simulations.

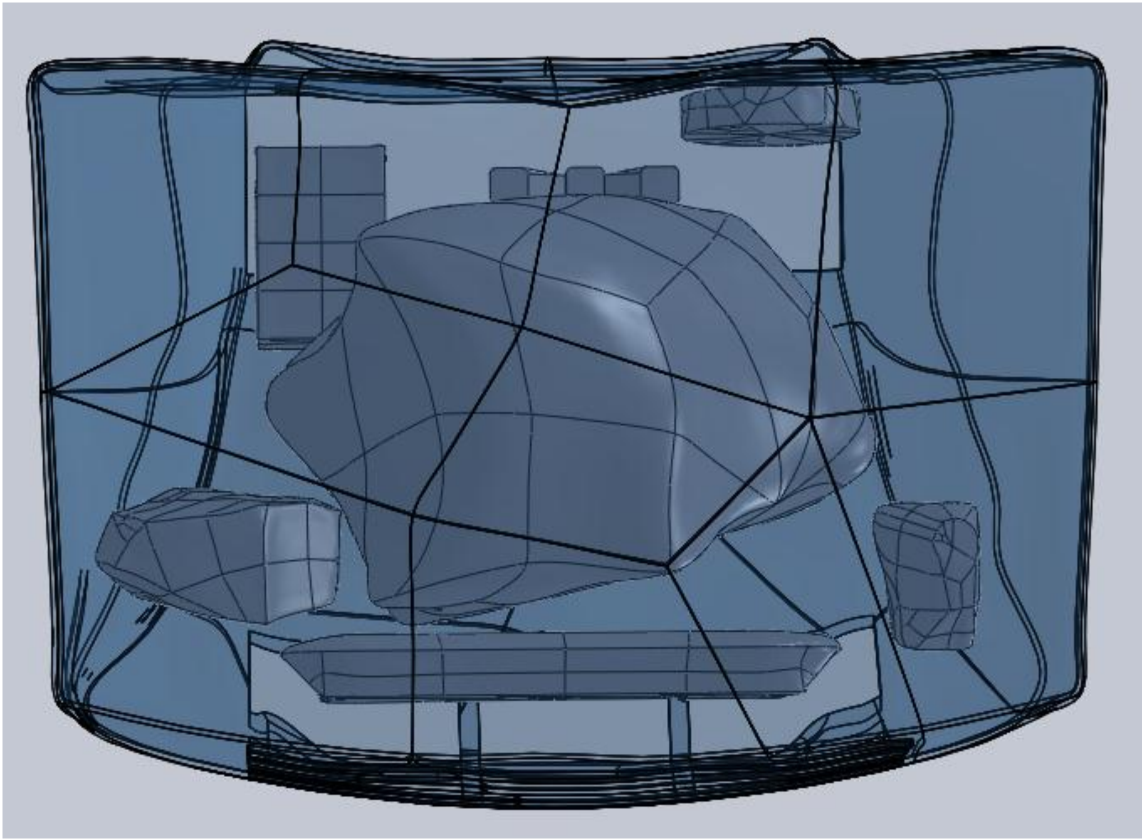


Figure 3.8: Top view of a 3-Dimensional CAD model of the car under hood, used for CFD simulations.

3.4 FLOW DOMAIN

After performing cleanup operations on the under hood geometry, the next concern is the selection of a flow domain with proper dimensions, for performing CFD simulations. The vehicle flow domain is an approximation of the real environment surrounding the vehicle. It helps the user to specify the boundary conditions which affect the air flow through the vehicle under hood. The inlet velocity of air into the flow domain specified by the user represents the vehicle speed.

The flow domain size has a great influence on the under hood air flow rates. The dimensions of the flow domain were determined according to the methodology derived and presented by Huang and Tzeng [8]. The distance from the bottom of the vehicle to the top of the vehicle was considered as the characteristic length, L_c. Huang and Tzeng [8], performed several CFD simulations with different flow domain sizes, and selected a flow domain with the dimensions as shown in the Figure 3.9. With the selected flow domain, the under hood air flow rates obtained showed a deviation of 0.81% from the experimental data. He thus concluded that the optimal flow domain should extend to five times the characteristic length form the sides, five times the characteristic length above the vehicle model, eight times the characteristic length in front of the vehicle model from the front grille area, and four times the characteristic length behind the vehicle model.

The flow domain for the underhood geometry was constructed, with dimensions as specified by Huang and Tzeng [8]. Figure 3.10 shows the under hood geometry with the recommended flow domain dimensions as presented by Huang and Tzeng [8]. Figure 3.10 displays the flow domain selected for performing CFD simulations over the underhood geometry.

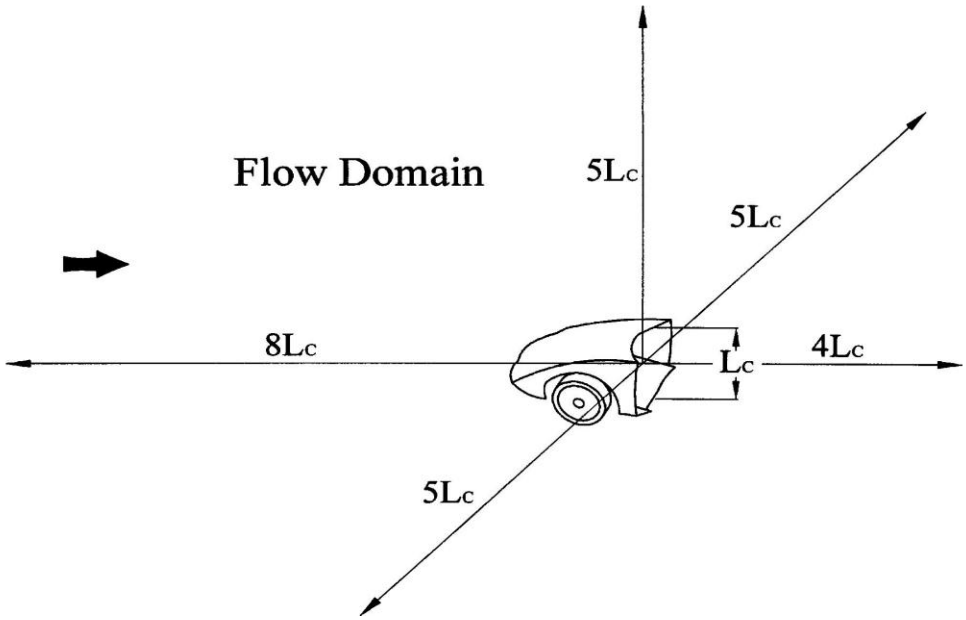


Figure 3.9: Recommended flow domain dimensions for the underhood geometry [8].

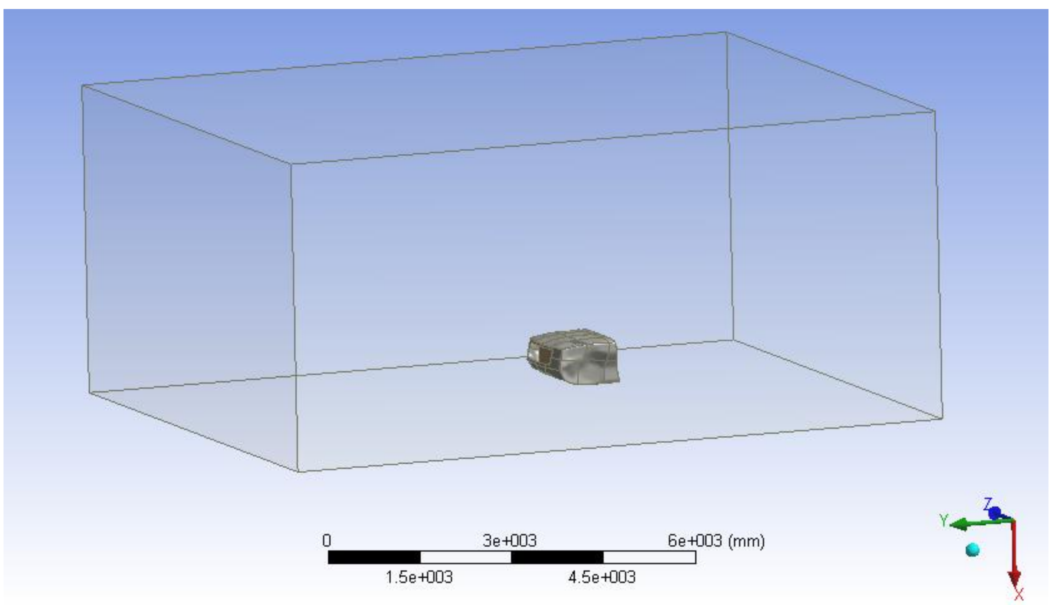


Figure 3.10: Flow domain used for performing CFD simulations on the car under hood geometry.

3.5 MESH GENERATION

The thus prepared CAD geometry in Ansys design modeler was then imported into the Ansys mesh setup cell. The project initially opted for the automatic meshing method available in Ansys mesh cell, before analyzing its quality. A mesh with good quality ensures faster convergence, increased accuracy in solution and a significant reduction in time taken to generate the mesh. Several mesh quality metrics are involved in order to quantify the quality of the mesh generated [14]. However the following three parameters are the primary metrics that quantify the quality of the mesh.

- Skewness
- Aspect ratio
- Smoothness

3.5.1 Skewness:

The following two methods are used to determine Skewness:

- Based on the equilateral volume deviation, Skewness is defined as the relative ratio between the optimal cell size and the actual cell size.

  $$\text{Skewness} = \frac{\text{Optimal cell size - cell size}}{\text{Optimal cell size}}$$

  This is applicable only to triangular and tetrahedral mesh methods. This is illustrated in Figure 3.11.

- Based on the normalized angle deviation, skewness is defined as the maximum value of the relative ratios between θ_e and θ_{min} obtained from all the cells.

$$\text{Skewness} = \max\left[\frac{\theta_e - \theta_{\min}}{\theta_e}\right]$$

Where,

θ_{min} = smallest angle of a cell.

θ_e = angle of the equilateral cell.

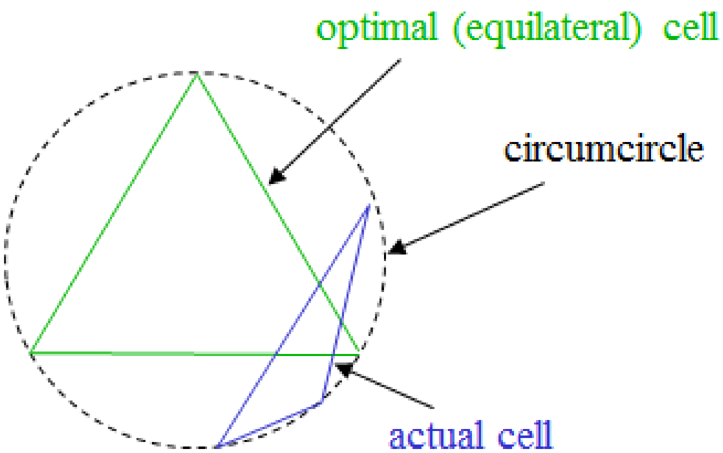


Figure 3.11: Figure displaying actual cell size and optimal cell size [14]

For hexahedral, triangular, quadrilateral mesh methods, skewness should be maintained below 0.8 and for the tetrahedral mesh method it should be below 0.9, to obtain a good quality mesh.

3.5.2 Aspect Ratio

Aspect ratio is defined as the ratio between the largest edge length and the shortest edge length of polygons [14]. It is equal to 1 (ideal) for an equilateral triangle or a square. Aspect ratio is illustrated in Figure 3.12. The aspect ratio of a good quality mesh should be kept below 40.

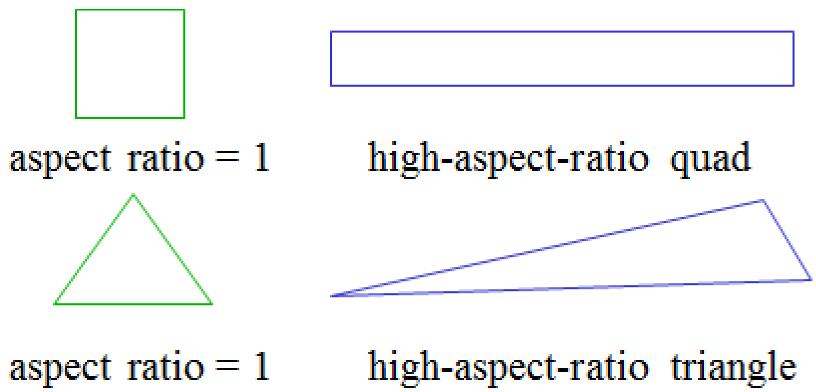


Figure 3.12: Illustration of aspect ratio [13].

3.5.3 Smoothness (Change in size)

The change in size from one cell to the adjacent cell should be gradual and not abrupt [14].

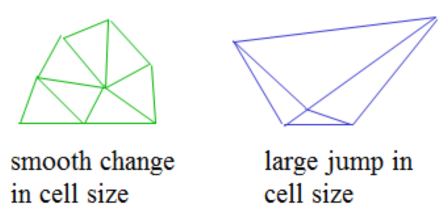


Figure 3.13: Illustration of smoothness [13].

Thus, the mesh metrics of the mesh obtained with automatic methods were checked to determine the cell skewness value. The maximum cell skewness value obtained with this method was close to 0.99 which is undesirable. The surface mesh obtained with the automatic method is shown in Figure 3.14.

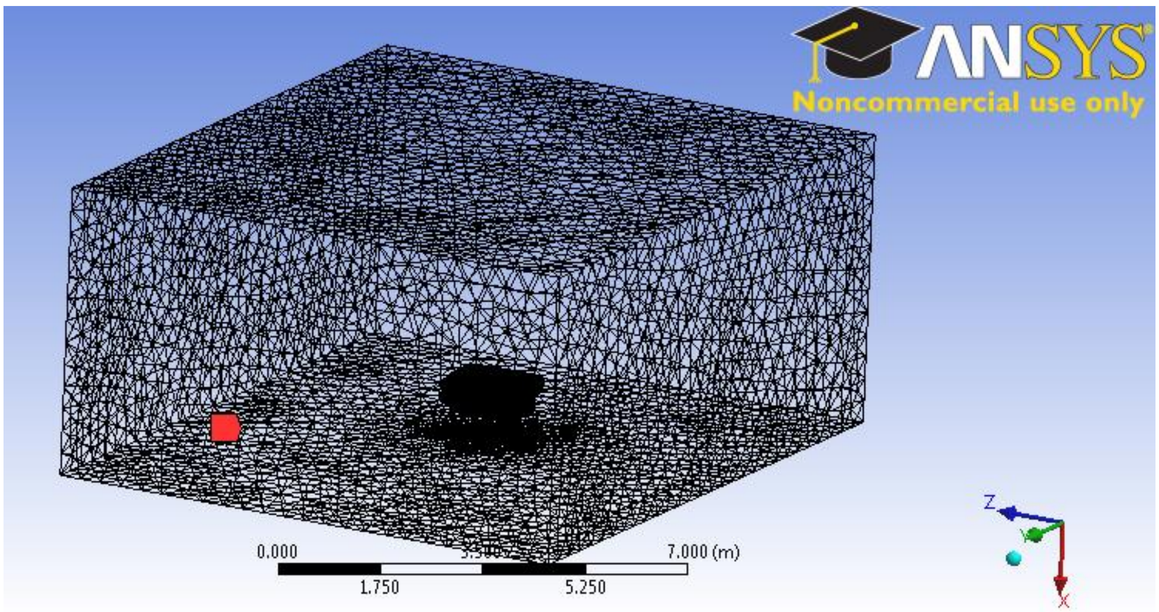


Figure 3.14: surface mesh generated on the flow domain with under hood geometry using automatic mesh methods.

The tetrahedral mesh method was then applied to generate the mesh on the entire geometry, considering its advantages over the automatic mesh method. The major advantageous attribute of the tetrahedral mesh method is that complex geometries can be meshed and added to the model at a faster pace, leading to more meaningful results [15].

Also, the transition from fine mesh regions (small components like the air filter) to coarse mesh regions (flow domain) is fast, which results in fewer cells overall. This reduces the overall computational power used.

The quality of the mesh obtained by employing the tetrahedral mesh method was better than that obtained by employing the automatic method. The tetrahedral mesh generated on the component air filter is shown in Figure 3.15. The mesh metrics obtained with it are also displayed, in Figure 3.15.

Though there was an improvement in the quality of the mesh obtained on the whole, the mesh metrics obtained were still not in the desirable range. Upon closer inspection, it was determined that the mesh generated on the component air filter alone was of poor quality, while the mesh applied on the rest had metrics in an acceptable range. The maximum skewness value of the mesh generated on the component air filter was 0.88, which is undesirable. Thus, to improve the quality of the mesh, different mesh methods were investigated. The automatic mesh method, applied to the component air filter alone resulted in a maximum skewness vale of 0.83. As discussed above, the automatic mesh method, when applied to all components did not yield a mesh with desirable quality, but when applied to air filter alone, the mesh generated on it was of good quality. So, for this project, the automatic mesh method was applied to generate the mesh on the air filter alone, while the rest were meshed using the patch conforming tetrahedral mesh method. Thus, the quality of the mesh on the whole was enhanced. The mesh generated on the component air filter by employing the automatic method is shown in Figure 3.16. The mesh metrics thus obtained is also displayed in Figure 3.16.

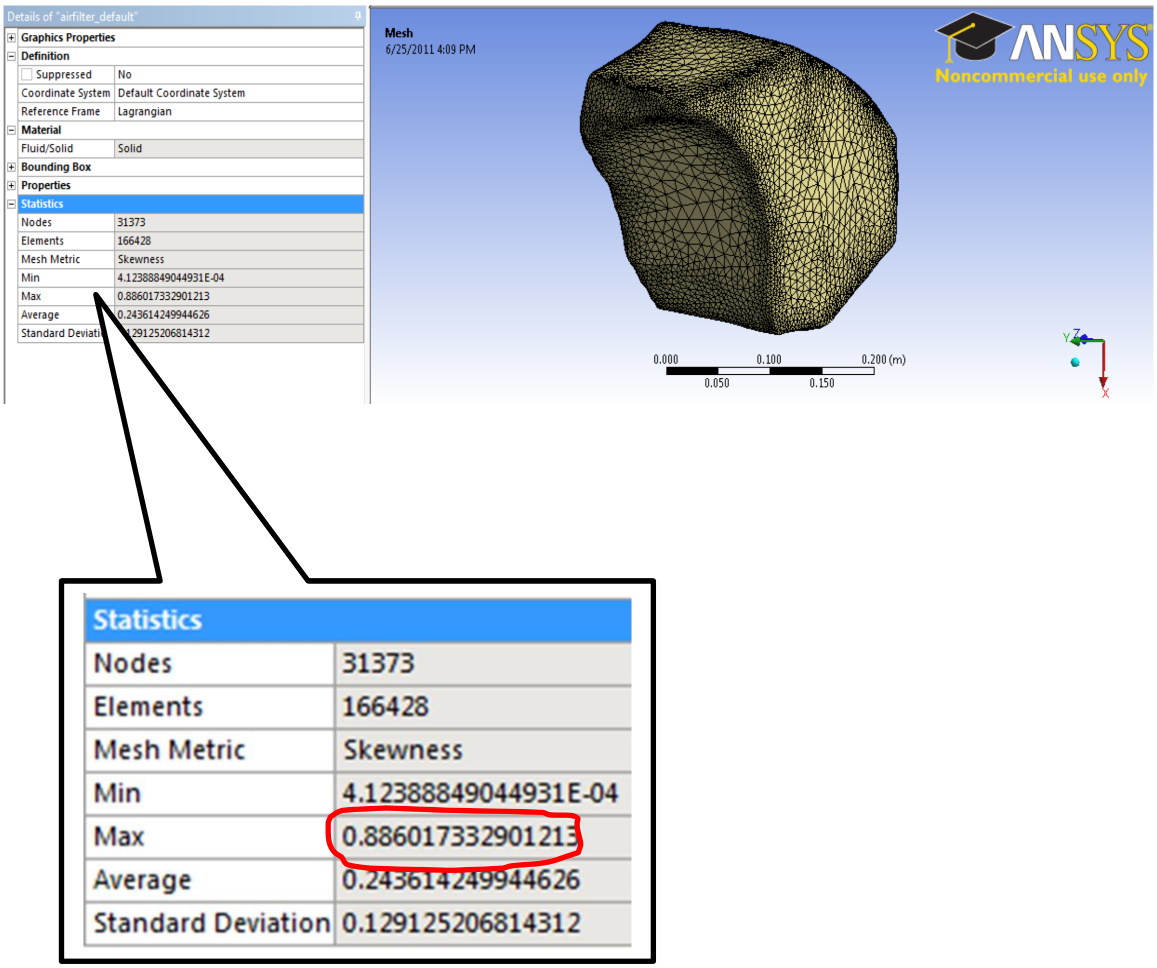


Figure 3.15: Mesh generated on air filter using tetrahedral mesh methods.

3.5.4 Named Selections

After a good quality mesh is generated on all parts, the next step is to create named selections. Named selections, help the user to identify the boundaries at which the boundary conditions are to be applied. Though, Ansys Fluent gives default names to all the boundaries generated in the design modeler, the user also has an option to specify its name.

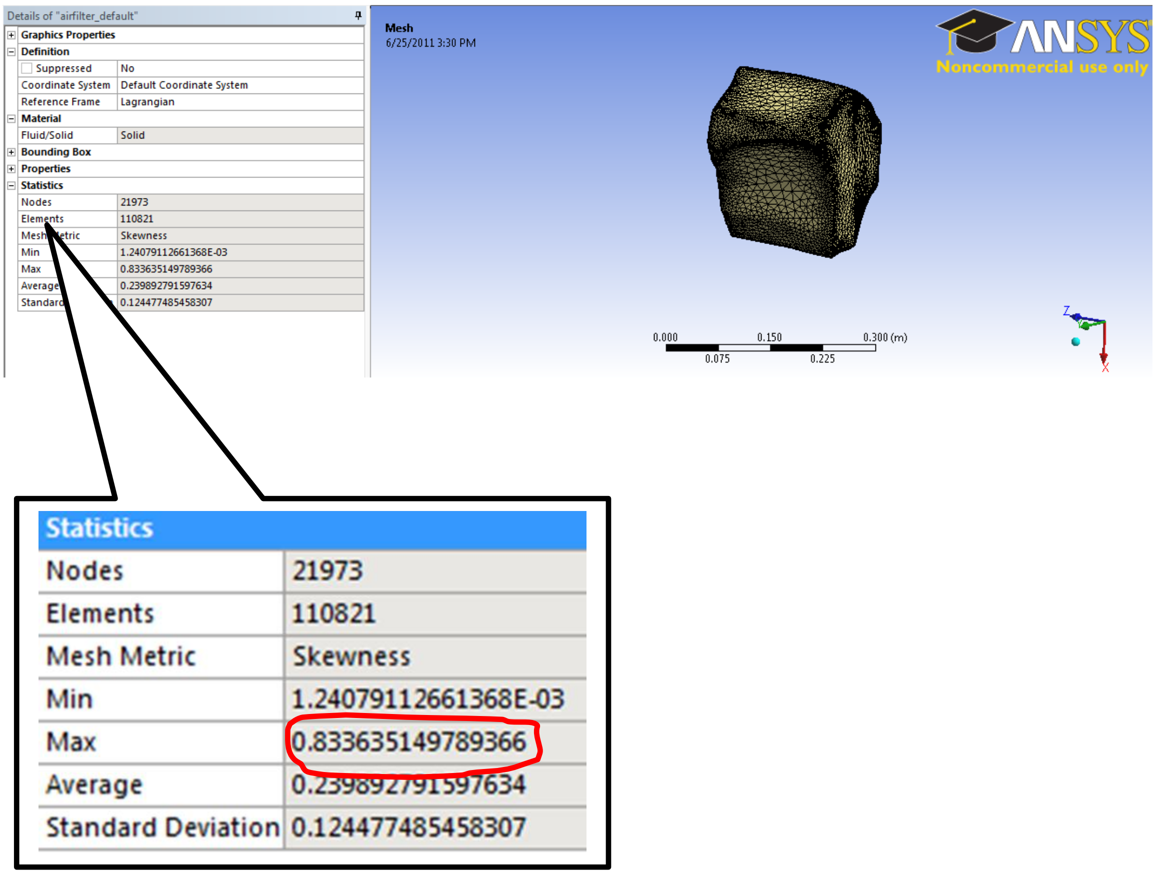


Figure 3.16: Mesh generated on air filter using automatic mesh methods.

In Ansys mesh cell, the user can use the face/body selection tools to specify the names of the boundaries/zones, which are of importance in the simulations. This improves the user's ability to recognize those boundaries and apply proper boundary conditions to them. Thus, the named selection enhances the ease in defining the boundary conditions in Fluent cell setup. Figures 3.17 and 3.18 shows the named selections used in this project.

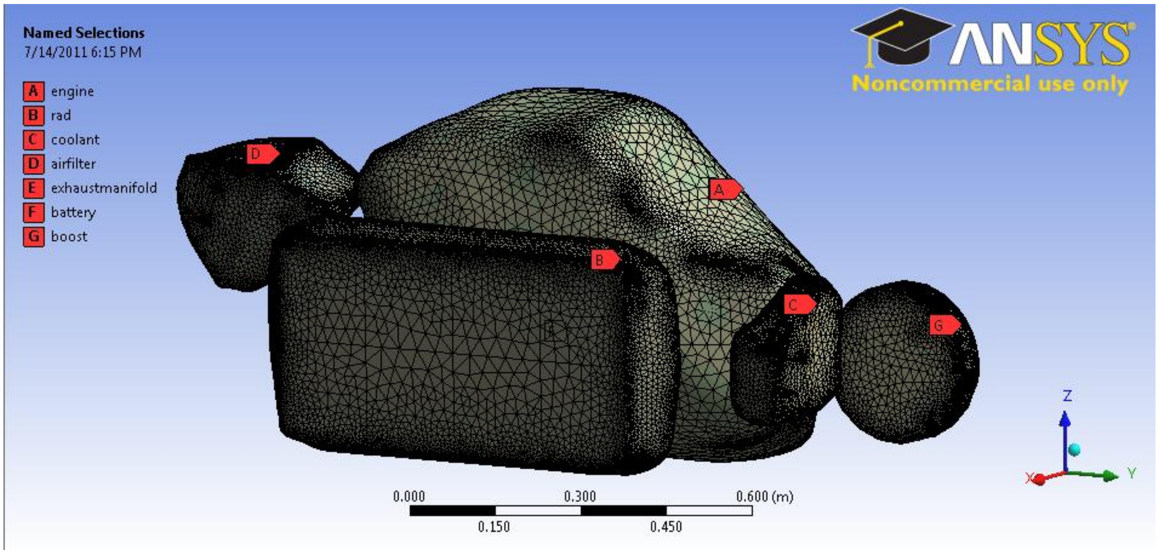


Figure 3.17: Named selections of the components inside the flow domain

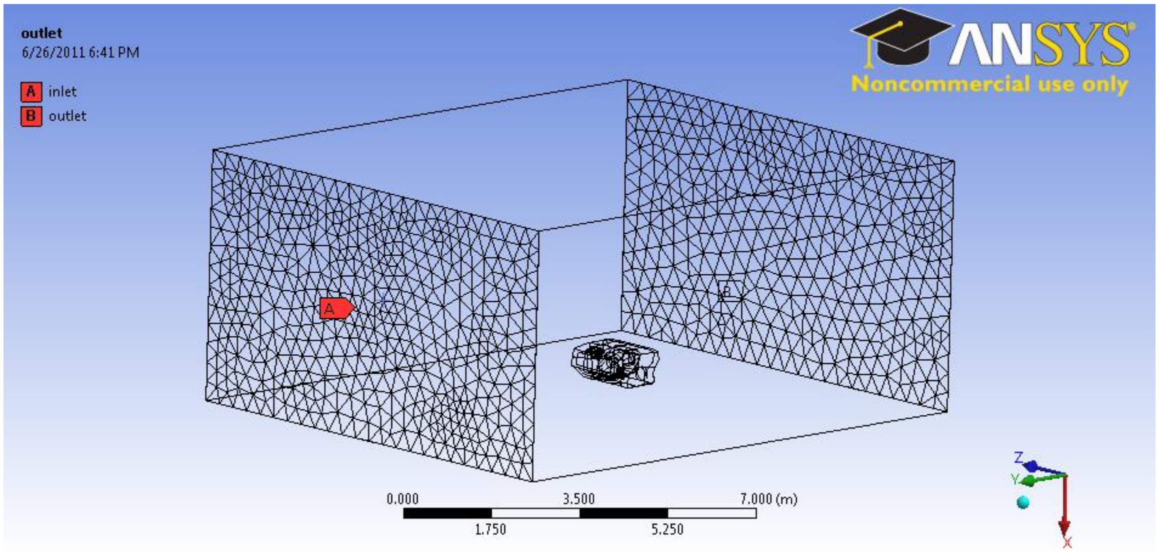


Figure 3.18: Named selections showing the inlet and outlet of the flow domain

CHAPTER FOUR

CFD SIMULATION

4.1 OVERVIEW

This chapter discusses the procedure followed in performing CFD simulations using the commercial CFD analysis tool, Ansys FLUENT over the components and in the domain specified in the previous chapter. The purpose of conducting these simulations was to predict air flow and thermal behavior of the vehicle under hood and thus predict the surface temperature of the components in the under hood compartment. A steady state flow and heat transfer analysis is employed in performing these simulations. Transient simulations were not included as they are computationally expensive and could be traded off, as our prime concern was to simulate the under hood thermal behavior at the most critical working conditions for optimization purposes.

The following major assumptions were made in the simulations performed

- Transient simulations were not included.
- Air was modeled as an incompressible fluid, as the simulations were conducted for lower air velocities.
- The velocity of air at the flow domain inlet was assumed to be 55 Km/h.
- Ambient temperature was assumed to be equal to 298.15 K.
- The radiator was modeled as an ungrouped macro heat exchanger model.
- The under hood compartment was considered to be adiabatic [2].

- Radiation heat transfer effects were assumed to be negligible [10].
- Constant surface temperature boundary condition is applied to both engine and exhaust manifold surfaces.

4.2 DETAILS OF INDIVIDUAL COMPONENT MODELING

The air flow pattern through the under hood compartment is very complex. Every component in the under hood compartment influences the flow pattern. They also influence the thermal behavior of the under hood compartment. Thus care has to be taken in modeling the components individually. Modeling of these components is explained in detail below.

4.2.1 Grille

A major aspect in the vehicle design process is the design of the front end grille. The airflow through the under hood compartment is primarily dependent on the grille design. Adequate care needs to be taken in modeling the grille to ensure that a proper amount of cooling air is let into the under hood compartment through the grille. Thus, an optimum inlet area is to be specified for the grille design, which ensures adequate air flow through the vehicle radiator core and thus meet the engine cooling requirements under all operating conditions. A schematic of the air flow through the under hood compartment is shown in Figure 4.1

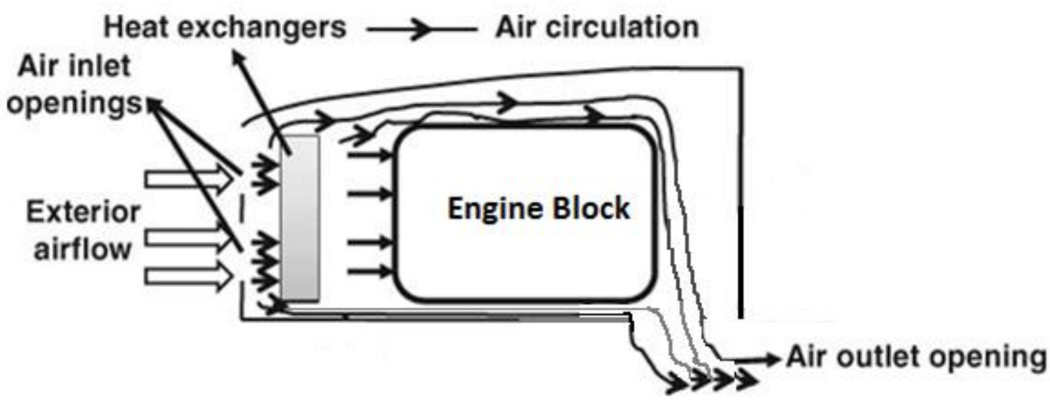


Figure 4.1: Schematic of the complex air flow through the under hood compartment [16].

Having considered the complexity in building up a 3-dimensional CAD model of the grille and its associated mesh, it was decided to model the grille as a porous medium [2]. But, with the advancements made in the Ansys design modeler and FLUENT capabilities, the surface geometry could be included along with the 3-dimensional CAD geometries. Thus, the under hood compartment with the grille was modeled accurately with the right amount of inlet area as a surface geometry. As mentioned above, the under hood compartment was assumed to be adiabatic and so its interactions with the surroundings need not be modeled. Finally, the under hood compartment with the grille was modeled as a thin surface model, which was not included in the meshing step [13].

4.2.2 Radiator

As mentioned above, proper regulation of air flow through the radiator core is very essential in the vehicle cooling process. The radiator acts as the heat sink for the heat generated from the engine. A coolant circuit travels from the engine compartment to the radiator carrying a part of the excessive heat generated by the engine. This heat is then rejected into the under hood environment as the air flows through the radiator's core.

Over the years, liquid-cooled radiators have become the most preferred radiators and are used in both passenger cars and heavy duty vehicles [2]. In vehicles employed with high performance engines, the cores of the radiator are modified as flat corrugated tubes to increase the area of heat transfer and thus increase the efficiency of the radiator. This further increases the complexity in modeling the radiator accurately in CFD simulations.

In CFD simulations, the radiator is modeled as a heat exchanger model, to predict the amount of heat rejected by the radiator to the under hood environment. It is also represented as a porous medium, to model the air flow through the radiator core. Viscous and inertial resistance required to model the radiator as a porous medium were calculated and applied [2].

The following two heat exchanger models are available in Ansys FLUENT

- Macro model
- Dual cell model

The Macro heat exchanger model either uses the number-of-transfer-unit (NTU) model or the simple effectiveness model for the heat transfer calculations. Whereas, the Dual cell model uses only the number-of-transfer-units model in performing the heat transfer calculations. The Dual cell model is more complicated and computationally expensive as it constructs the solution of the auxiliary fluid flow on a separate mesh, unlike the Macro model, where the flow is modeled as a 1-D flow. These models are used to compute the auxiliary fluid inlet temperature when the amount of heat rejection is fixed and known or to compute the total heat rejection when the fixed auxiliary fluid inlet temperature is known.

In this project, an ungrouped Macro heat exchanger model from Ansys FLUENT was used to model the radiator. In a standard heat exchanger core, the auxiliary fluid (coolant) temperature is not constant throughout its flow path, along the direction of the auxiliary fluid flow. As a result, heat rejection from the radiator is also not constant over the entire core. Thus to incorporate this uneven distribution of the heat rejection in the heat transfer calculations, the control volume representing the heat exchanger core is divided into macros along the auxiliary fluid path as shown in Figure 4.2. In the Figure shown the heat exchanger core is discretized into 2×4×2 macros. This implies that the auxiliary fluid flows through the heat exchanger core in two passes; each pass is then divided into four rows and two columns of macros. Now, the auxiliary fluid inlet temperature to each macro is computed and then used in the heat transfer calculations. This approach of modeling the heat exchanger core by discretizing it into smaller sub domains called macros provides more realistic solutions for the heat rejection calculations according to [18].

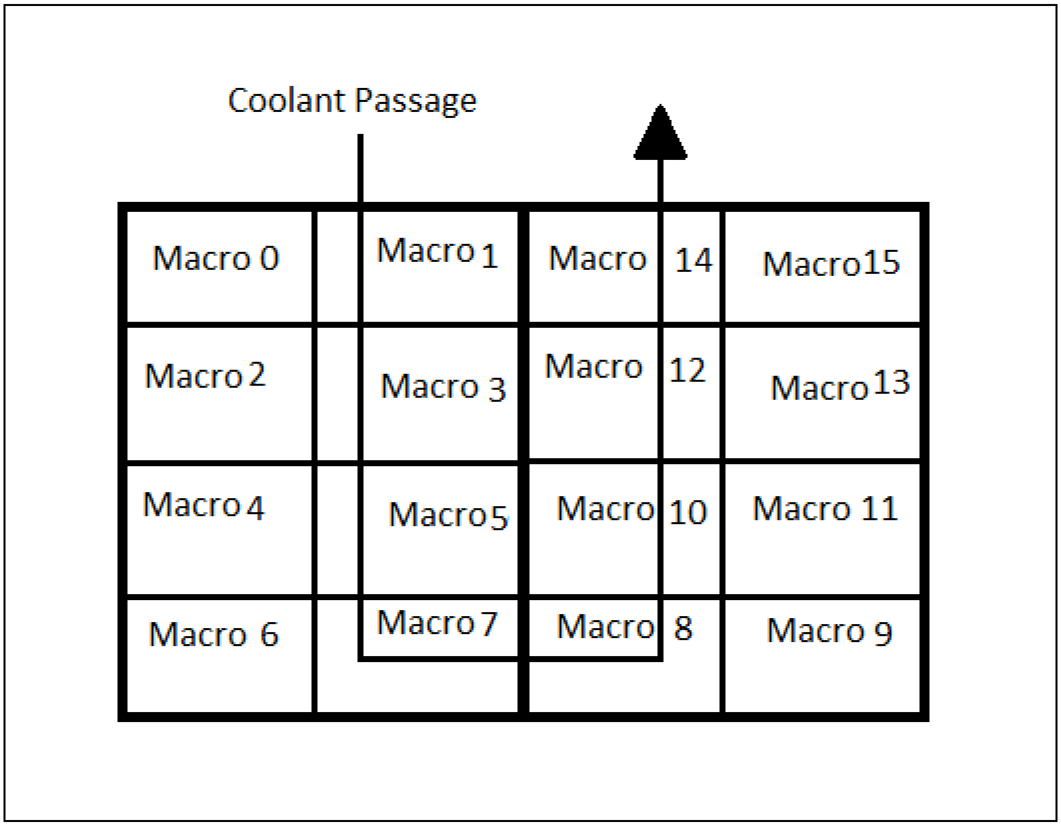


Figure 4.2: Heat exchanger core divided into 2× 4×2 macros [18].

Having discussed the ease in setting up the problem using the macro heat exchanger model, there are also a few restrictions with using this model which have to be considered. The major limitations in using this model are as follows [18]

- The heat exchanger core must approximately be rectangular in shape.

- The direction of flow of the primary fluid must be aligned with one of the three orthogonal axes defined by the heat exchanger core.
- The change in phase of the coolant fluid cannot be modeled and the fluid flow is assumed to be 1-D.

In this project, the radiator modeled as an ungrouped macro heat exchanger model was discretized into the default 2×5×1 number of macros specified in Fluent. These macros are identified in this model with different colors, as shown in the Figure 4.3. The radiator core model given for this project was approximately rectangular as shown in the Figure below. The direction of flow of the primary fluid is aligned with the x-axis.

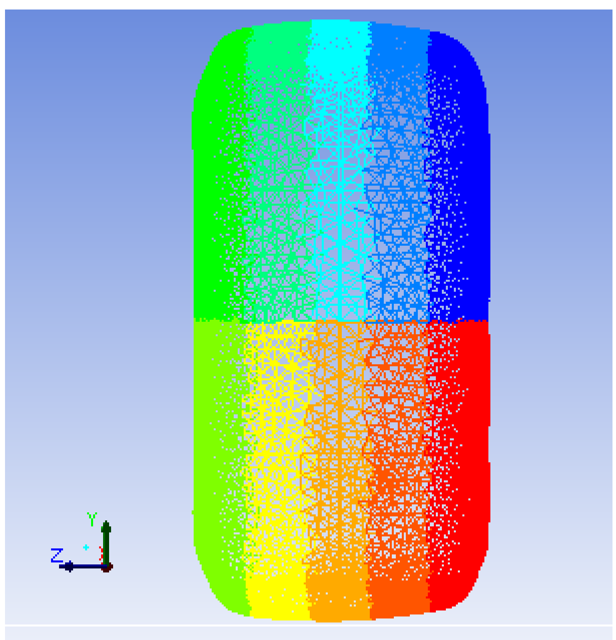


Figure 4.3: Meshed CAD model of the Radiator displaying the macros.

As mentioned above, the radiator was modeled using the ungrouped macro heat exchanger model in Ansys FLUENT. To do so, the following heat transfer data was required.

- The fluid zone representing the heat exchanger core had to be specified. In this project, the radiator modeled as a rectangular core represented the fluid zone.
- Either the fixed heat rejection or the coolant inlet temperature had to be specified and in this project the fixed inlet temperature of the coolant was specified as 358.15 k.
- The primary fluid temperature also was specified as 298.15 k.
- Then, the “Heat exchanger performance data” and the “Core porosity model” had to be specified.

In this project, the “Heat exchanger performance data” required was obtained from the literature. Dohoy et al. [17] had developed a numerical model using the finite difference method based on the thermal resistance concept to predict the effect of the design parameters on the heat exchanger performance. For the validation of the model, the heat rejection performance of a typical car radiator was simulated for different air flow rates and coolant flow rates. This data was then validated with the experimental data provided by the manufacturer. The experimental data was presented in the paper as a plot of the heat rejections corresponding to different air flow rates and coolant flow rates. The data used in this project is shown below in Figure 4.4. In the Figure shown, heat rejection of the radiator was plotted corresponding to three coolant flow rates and three air flow rates.

The experimental data plotted, was represented with symbols, whereas, the simulation data was represented with the dotted lines as shown in the Figure 4.4

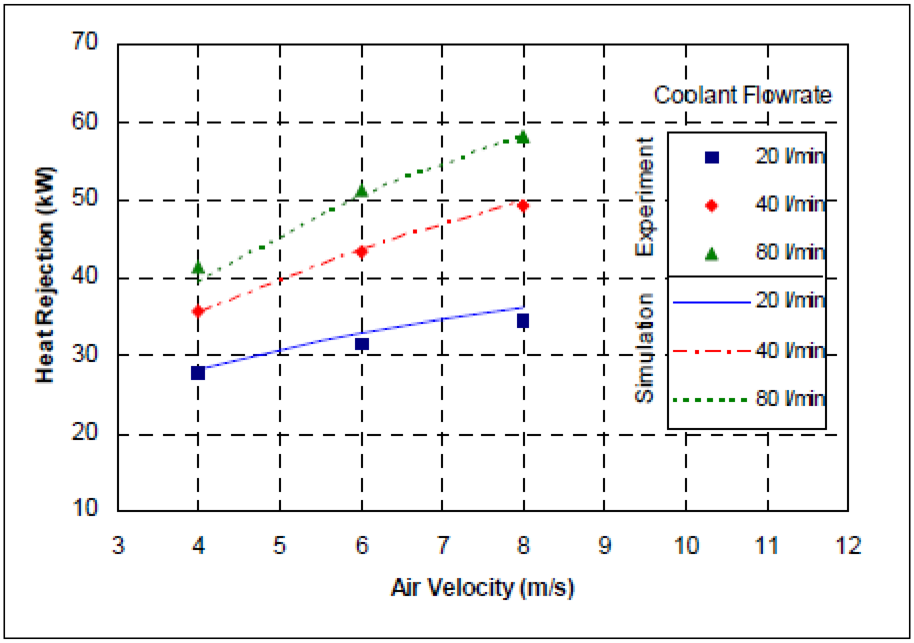


Figure 4.4: Heat exchanger performance data [17].

Thus the heat transfer data Table required to model the radiator was built with the data inferred from the above Figure. The screen shot of the Heat transfer data Table used in this project is presented in Figure 4.5. The heat rejection of the radiator in watts corresponding to three different flow rates of the primary fluid flow and the auxiliary fluid flow was specified in this Table.

Next needed is the data required to set up the core porosity model in the ungrouped macro heat exchanger model. This data was calculated from the experimental data provided for this project. The experimental data provided was the pressure drop obtained across the radiator for different flow rates, which is shown in Table 4.1.

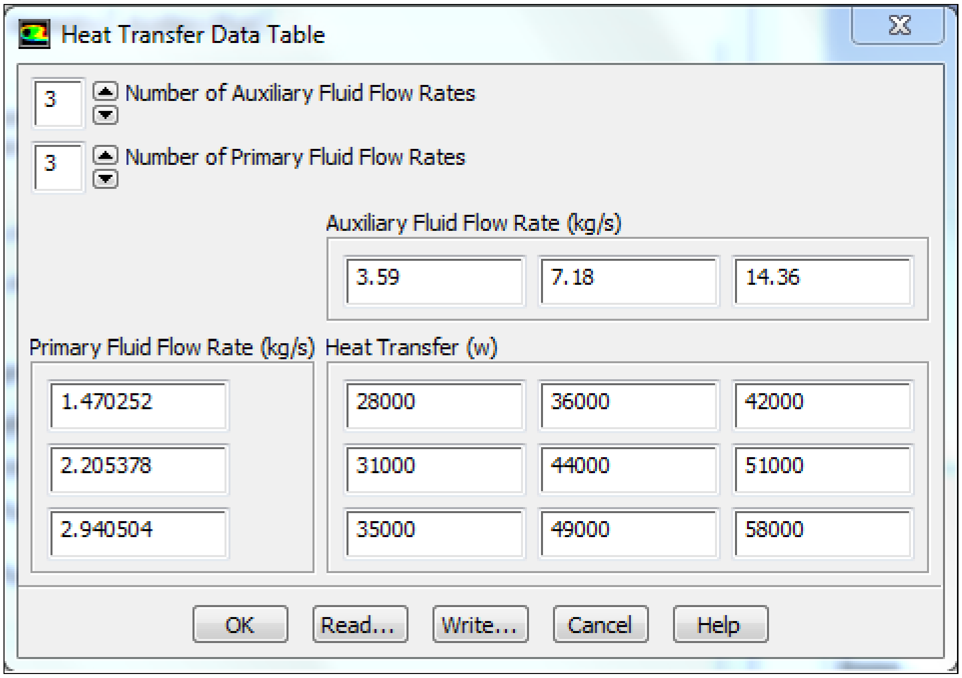


Figure 4.5: Heat transfer data Table in FLUENT.

Table 4.1: Experimental Data for the pressure drop across the Radiator

Velocity (m/s)	Pressure Drop (Pa)
0.510204	44
1.020408	114
1.530612	210
2.040816	331
2.55102	477
3.061224	650
3.571429	849
4.081633	1073
4.591837	1324
5.102041	1599

With the given experimental data, an xy curve is plotted as shown in Figure 4.6. Then a trendline passing through the given data points was created, which yielded the following equation.

$$\Delta \mathrm{p} = 4.9691v^2 - 61.0355v + 1.88$$

Where, $\Delta \mathrm{p}$ is the pressure drop across the radiator and v is the velocity of the primary fluid through the radiator.

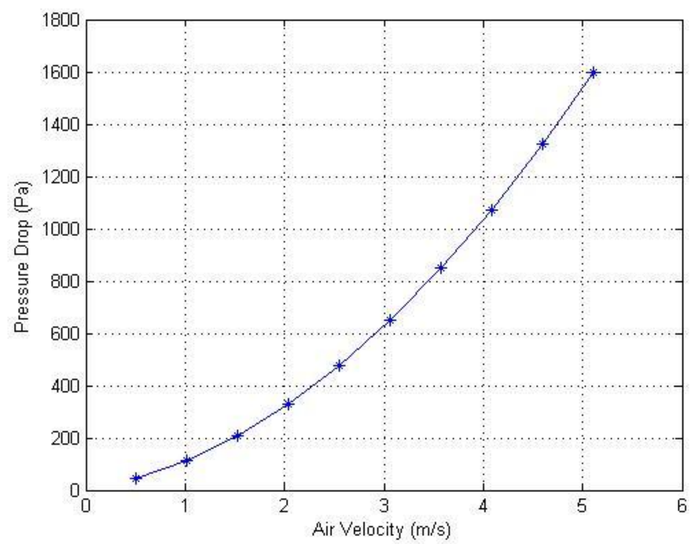


Figure 4.6: Pressure drop characteristics of the Radiator.

In Ansys FLUENT, the porous medium is modeled by the addition of a momentum source term to the fluid flow equations. In the case of a simple homogeneous porous medium the source term is defined as shown below [19].

$$S_i = -\left(\frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v| v_i \right)$$

Where μ (dynamic viscosity) = 1.7894×10^{-5} Pa.s, ρ (density) = 1.225 Kg/m^3, C_2 = inertial resistance factor, $\frac{1}{\alpha}$ = viscous resistance factor.

Thus by comparing the above two equations, the inertial resistance factor C_2, and the viscous resistance factor $\frac{1}{\alpha}$ required for setting up the core porosity model in the ungrouped heat exchanger model, were calculated and applied to the porous model in Ansys FLUENT . A screen shot of the porous model inputs is shown in the Figure 4.7.

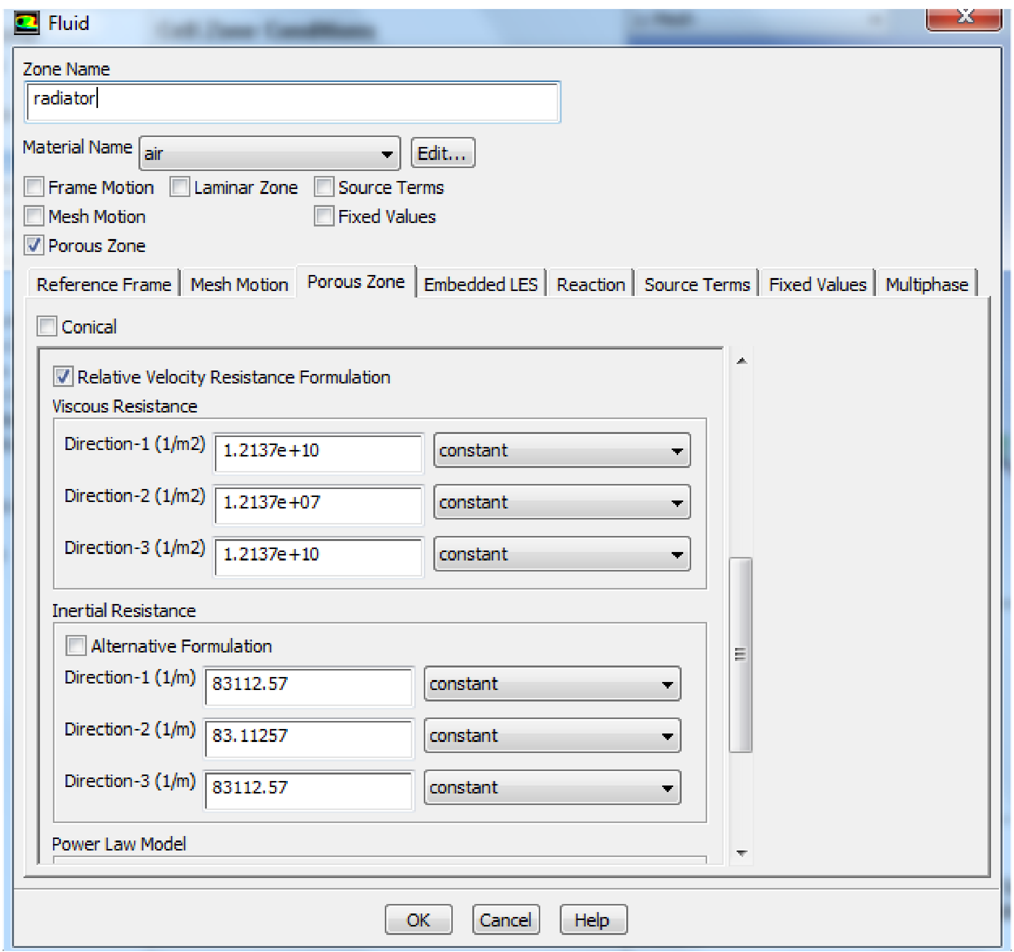


Figure 4.7: Porous model inputs for Radiator in Ansys FLUENT 13.0

4.2.3 Engine and Exhaust Manifold

The engine is the prime source of energy for any vehicle. Fuel combustion inside the engine compartment liberates heat energy, which is converted into mechanical energy. The thus generated mechanical energy is delivered to the drive wheels through the vehicle power train and causes the vehicular motion. In typical vehicles, the energy generated through the combustion of the fuel inside the engine compartment is dispersed into three prime areas. Approximately, only about 40% of this energy is converted into mechanical energy and goes to driving the wheels as discussed above. 30% of the combustion energy leaves through the exhaust and heat rejections to the under hood environment. The rest 30% of the energy is carried away by the coolant system and is then rejected to the under hood environment through the heat exchangers, i.e. radiator [20]. The above discussed breakdown of the thermal energy generated into the three areas is approximate and can vary significantly when the parasitic losses by components such as fans and pumps are taken into consideration.

Thus, one can assume that approximately 20% of the thermal energy generated in the engine compartment is rejected to its surroundings. Also, with the tightly packed under hood environment surrounding the engine compartment, the heat rejected to the surroundings can be critical for any heat sensitive components in its near vicinity. The airflow past the engine acts as a thermal sink for this heat load. Thus proper regulation of the air flow around the engine compartment is crucial.

In CFD simulations, the engine surface is modeled as a non-slip wall. An isothermal constant temperature boundary condition is applied to the engine surface to account for its heat rejection to the under hood environment.

As discussed above, 30% of the combustion energy released leaves the engine compartment through the exhaust system. Thus, the outer surface of the exhaust system is another primary source of heat into the under hood environment. In CFD simulations, the exhaust manifold's surface is again modeled as a non-slip wall and an isothermal constant temperature boundary condition is applied to this surface.

form the results presented in Winnard et al. [10] it can be inferred that the amount of heat transferred from exhaust manifold constitutes only a small fraction of the total amount of heat transferred to the under hood environment. The mode of heat transfer prevalent for this heat transfer is primarily radiation. All other components under the hood primarily transfer heat through convection. Thus, on the basis of the inference made, the radiation effects can be neglected to simplify the thermal simulations over the under hood environment. Thus, in this project the above mentioned assumption of neglecting the radiation effects is made.

4.3 BOUNDARY CONDITIONS AND CELL ZONE CONDITIONS

The Cell zone and boundary conditions specify the flow and thermal variables on the boundaries of the physical model. These cell zone and boundary conditions, thus, provide the required information to describe the air flow and thermal behavior inside the model. They are, therefore, a critical component in building up the models for the CFD simulations. Thus, adequate care needs to be taken in specifying the appropriate cell zone and boundary conditions [19].

The boundary types available in Ansys FLUENT are classified as

- Flow inlet and exit boundaries: All the different types of boundary conditions that can be applied on the boundaries through which the fluid flow either enters or exits falls into this category. These boundaries can be specified with the following boundary conditions.
    - Pressure inlet
    - Velocity inlet
    - Mass flow inlet
    - Inlet vent
    - Intake fan
    - Pressure outlet
    - Pressure far-field
    - Outflow
    - Outlet vent
    - Exhaust fan
- Wall boundaries: These are the boundary conditions that are applied over the surface of the components inside the model. Available boundary conditions in this category are

- ➢ Wall type
- ➢ Symmetry type
- ➢ Periodic type
- ➢ Axis

- Internal face boundaries: The surface boundaries through which both the energy and mass transfer can occur inside the model fall into this category of boundaries. Available boundary conditions in this category are
    - ➢ Fan
    - ➢ Radiator
    - ➢ Porous jump
    - ➢ Interior
    - ➢ Wall

Cell zones in FLUENT are specified as fluids and solids. The porous medium in FLUENT is treated as a fluid zone and is specified in the cell zone conditions. The boundary conditions applied to the simulations done in this project are presented in Table 4.2.

Table 4.2: Boundary conditions applied to the simulations in this project

Component	Boundary type	Properties
Flow domain inlet	Velocity-inlet	16.67 m/s
Flow domain outlet	Pressure-outlet	101.325 pa
Engine surface	Non-slip wall	600 k
Exhaust manifold surface	Non-slip wall	750 k

In this project, the air flow through the under hood was considered to be turbulent. To model the turbulence, the standard k-ε turbulence model available in Ansys FLUENT was used. The turbulence of the incoming air was specified using the turbulent intensity and hydraulic diameter. The turbulent intensity was assumed to be 25% with a hydraulic diameter of 5.88 m.

4.4 SOLUTION ALGORITHMS AND UNDER RELAXATION PARAMETERS

Ansys FLUENT provides the user with the following two numerical methods to perform the simulations and determine the velocity and temperature fields.

- Pressure-based solver
- Density-based solver

Traditionally speaking, the pressure-based solvers were developed for solving problems that deal with low speed incompressible flows. Whereas, the density-based solvers were developed for solving problems that deal with high speed compressible flows [17]. In recent years, the advancements were made in both solvers to solve and operate for a wide range of the flow conditions.

Using both methods, Ansys FLUENT solves the governing equations for the conservation of mass, momentum, energy, and other scalars such as turbulence etc. To do so, a control-volume based technique is used by both solvers [18]. The control-volume technique used consists of:

- Discretizing the domain into smaller control volumes by generating a computational mesh.
- Solving all the governing equations on the individual control volumes and then integrating them all together to determine the variables such as velocities, pressure, temperature and other scalars.

Both methods employ a finite volume based discretization procedure, but the approach used to solve the discretized equations is different.

To summarize, we assumed that the air flow through the under hood geometry was incompressible and the velocity of the flow at the inlet was not very high. Thus, in this project, a pressure based solver "SIMPLE", available in Ansys FLUENT, is used for performing the CFD simulations over the complex under hood geometry.

The under-relaxation parameter reduces the amount of correction applied to the solution. Thus, the solution is updated in smaller steps, resulting in an increased time for the solution to achieve convergence. This in turn increases the stability of the solution achieved. Thus a proper set of the under-relaxation parameters need to be chosen to achieve a solution with desired accuracy and stability. The set of under-relaxation parameters used in this project is presented in Table 4.3. Finally, in this project, a linear upward differencing scheme is applied in approximating the derivatives involved. Linear upward differencing scheme provides a good compromise between stability and accuracy of the solution obtained [18].

Table 4.3: Under-Relaxation parameters

Parameter Name	Under-Relaxation Parameter Value
Pressure	0.3
Density	0.7
Body Forces	0.7
Momentum	0.5
Turbulent Kinetic Energy	0.6
Turbulent Dissipation Rate	0.6
Turbulent Viscosity	0.7
Energy	0.7

CHAPTER FIVE

RESULTS

A CFD simulation with the solver parameters and boundary conditions as specified in the previous chapter is performed to provide the user with the fluid velocity, temperature and pressure values through the solution domain for problems that involve complex geometries. In a typical under hood CFD simulation of any vehicle, contour and vector plots for velocity and temperature over different planes in the solution domain are plotted. From the results thus obtained, the need for relocating the components to prevent the under hood thermal environment form being hostile, is assessed.

In this project, CFD simulation over the car under hood region has been setup and run to predict the velocity and temperature distribution. Also, several CFD simulations were setup by relocating the components in the under hood region, to assess the change in the underhood thermal behavior. The results thus obtained, were used to assert the importance of packaging considerations in restraining the under hood thermal behavior to a safe limit.

In this section, the contour and vector plots for velocity and temperature fields obtained from the CFD simulation of the car under hood are presented. Figure 5.1, shows the contour plot for velocity, on a plane cut through the engine and exhaust manifold.

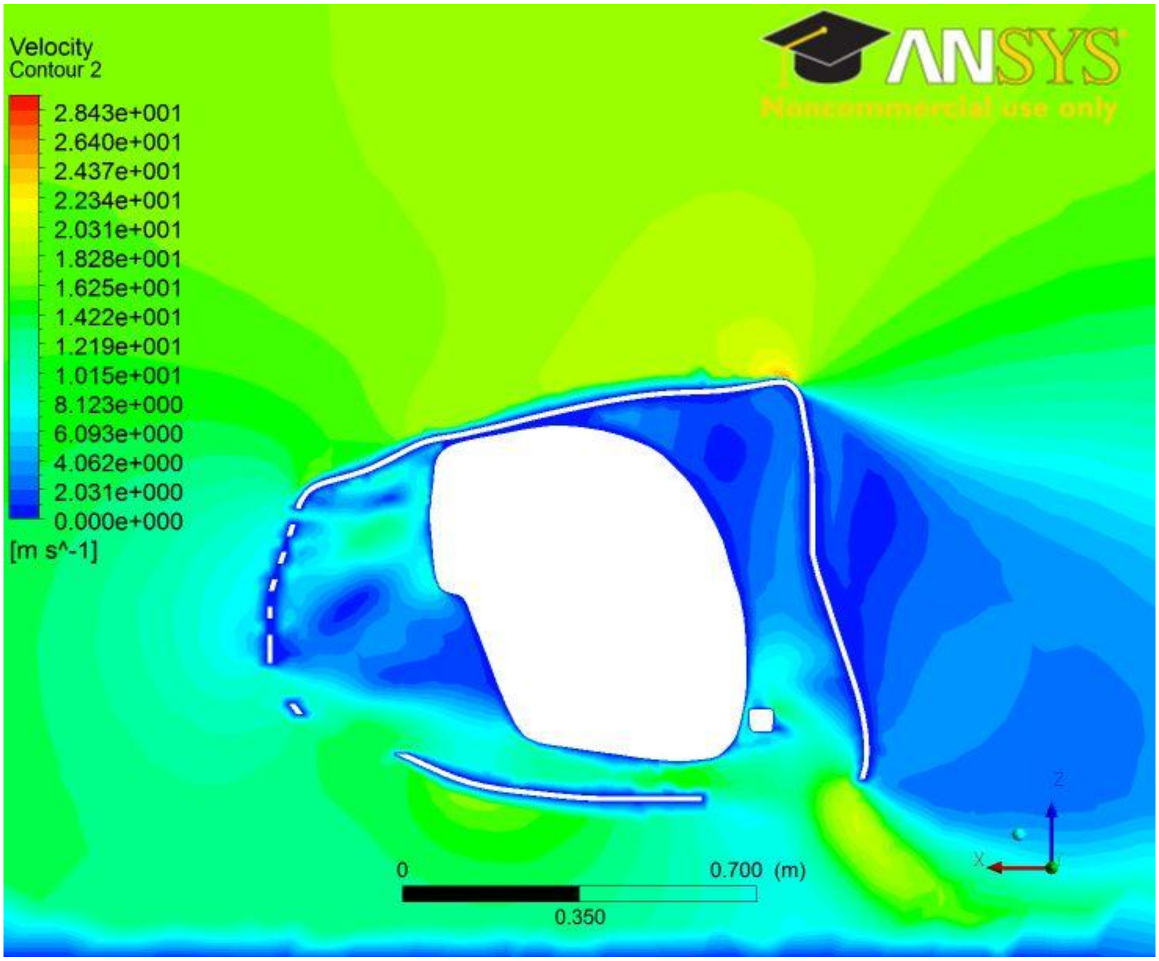


Figure 5.1: Contours of velocity on a plane through the engine and the exhaust manifold

The velocity contour plot shown in the Figure 5.1 indicates that the air velocity is restricted along the boundaries of the engine and also in few zones surrounding the engine. This is seen in the upper right corner of the contour plot presented. These zones, where the air velocity is restricted, stand a chance for the risk of becoming the pockets of high temperature zones in the under hood region.

The streamlines launched from the flow domian inlet, drawn over the plane passing throught the radiator, engine, exhaust manifold and the brake booster, are shown in the Figure 5.2. This Figure shows the formation of the recirculation zones in and around the engine vicinity. These recirculation zones act as obstacles to the air flow and thus create regions over which the air velocity is restricted. The resulting regions of restricted air velocity matched those obtained in the vector plot of the velocity field presented in Figure 5.1.

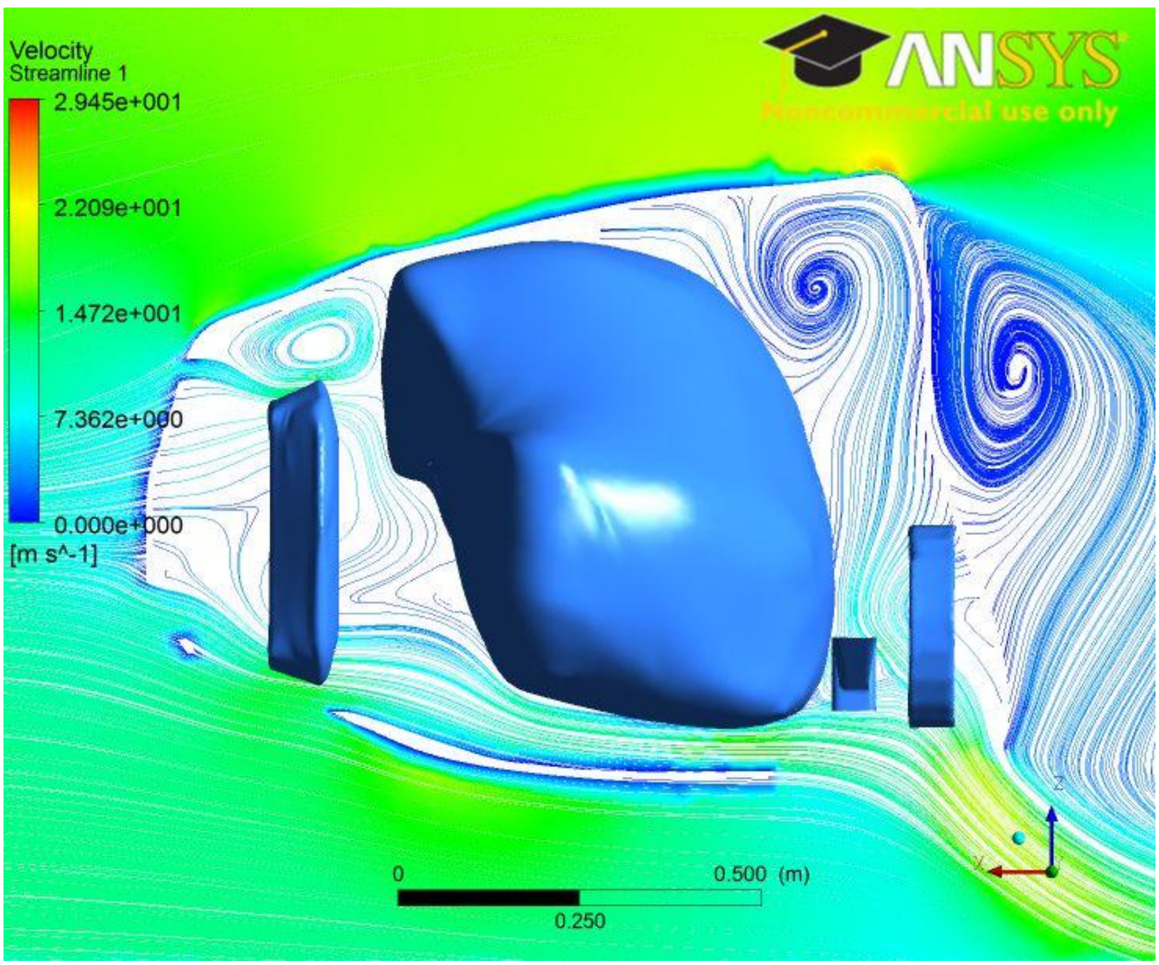


Figure 5.2: Streamlines launched from the domain inlet, drawn over the plane passing throught the radiator, engine, exhaust manifold and the brake booster.

The vector plot for the velocity field on the same plane that passes through the radiator, engine, exhaust manifold and the brake booster is also shown in the Figure 5.3. The same recicrculation zones can be seen in the vector plot of the velocity filed as well. This ascertains the presence of the recirculation zones in the under hood region of the car.

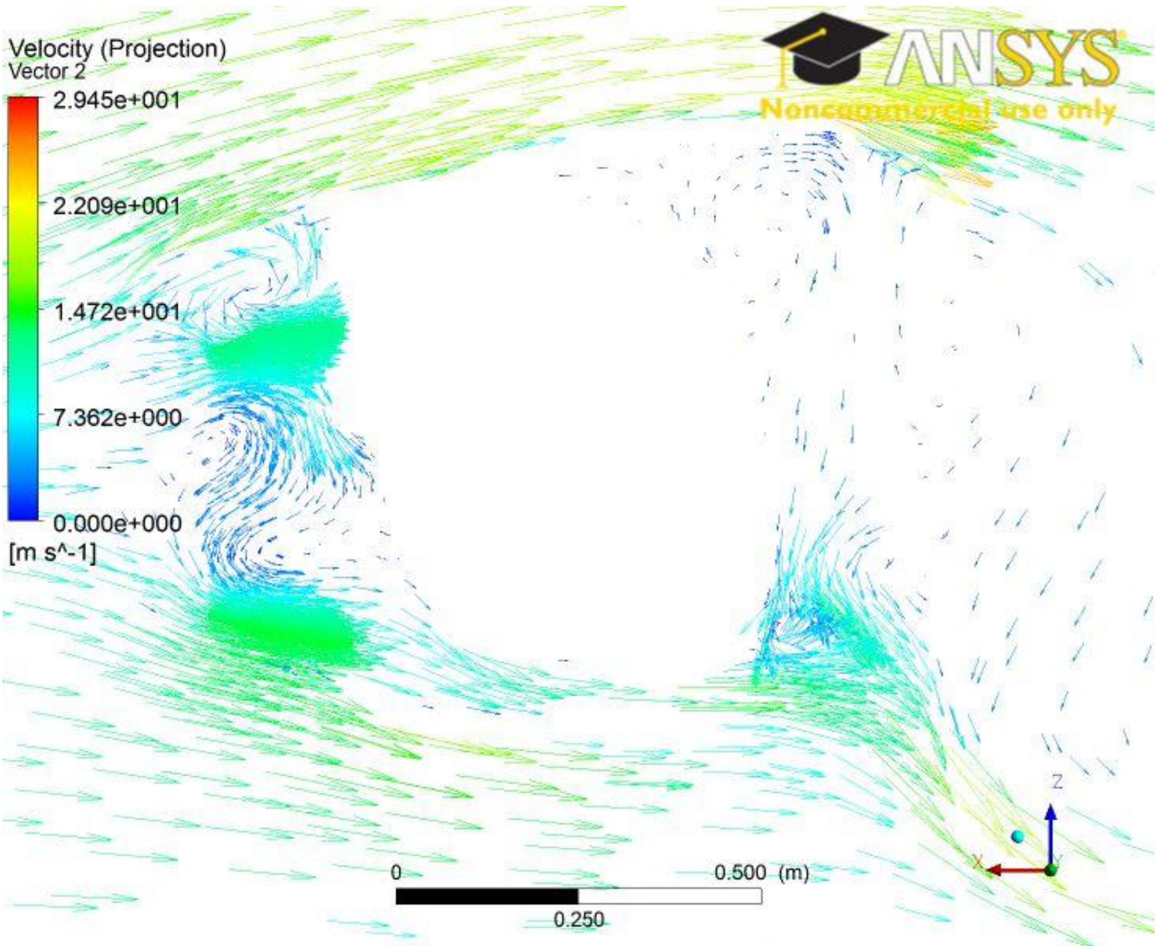


Figure 5.3: Vector plot for the velocity field on the plane that passes through the radiator, engine, exhaust manifold and the brake booster

As discussed above, in the zones where the air velocity was restricted indicated the presence of high temperature zones. The temperature contour plot for the same plane passing through engine and exhaust manifold is shown in the Figure 5.4

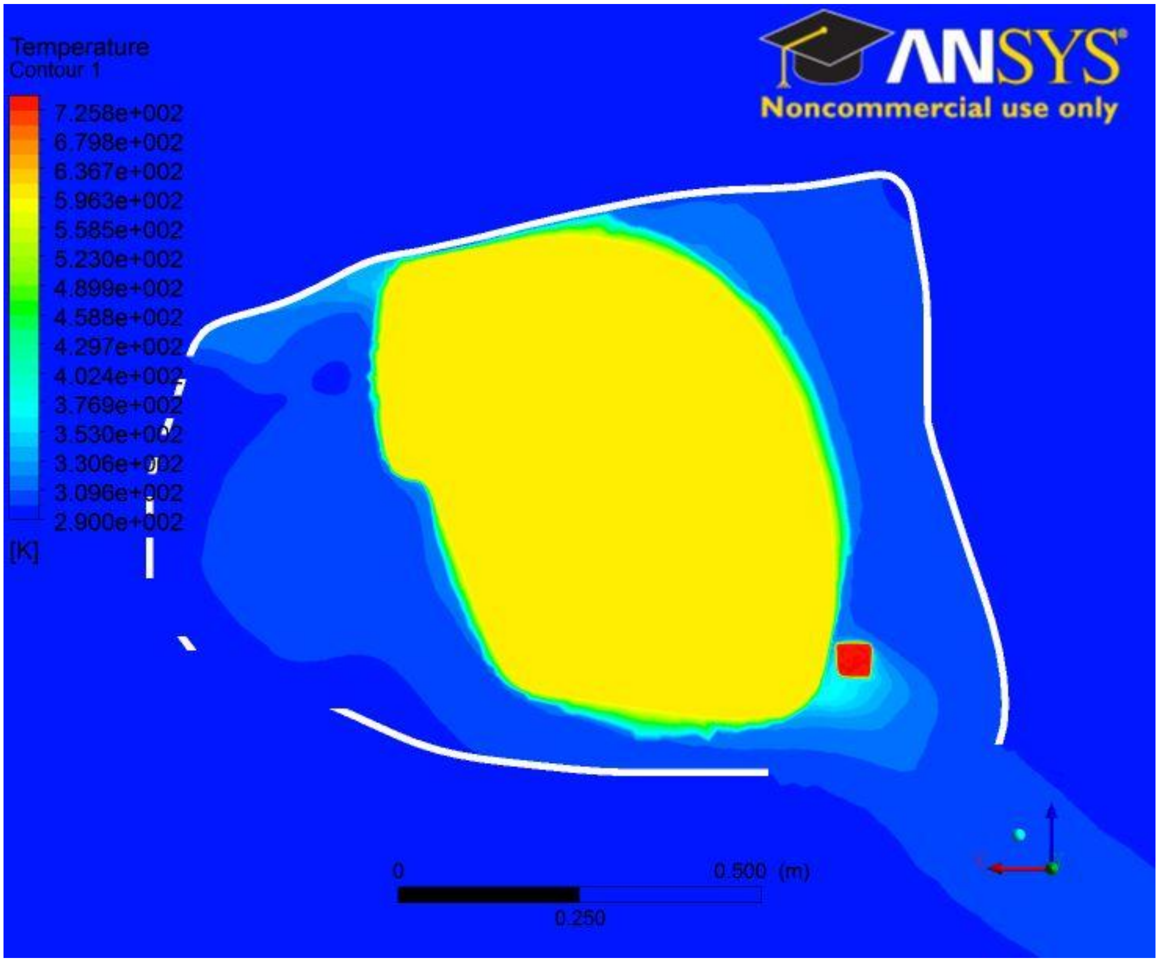


Figure 5.4: Contours of temperature on a plane through the engine and the exhaust manifold

It can be seen from the temperature plots, that the air trapped in the vicinity of high temperature sources such as engine and exhaust manifold, absorbs heat from the component walls. But, this absorbed heat is not being carried away, due to the poor air circulation observed in the under hood region. Figure 5.5 shows a temperature plot on a horizontal plane passing through the radiator and the engine

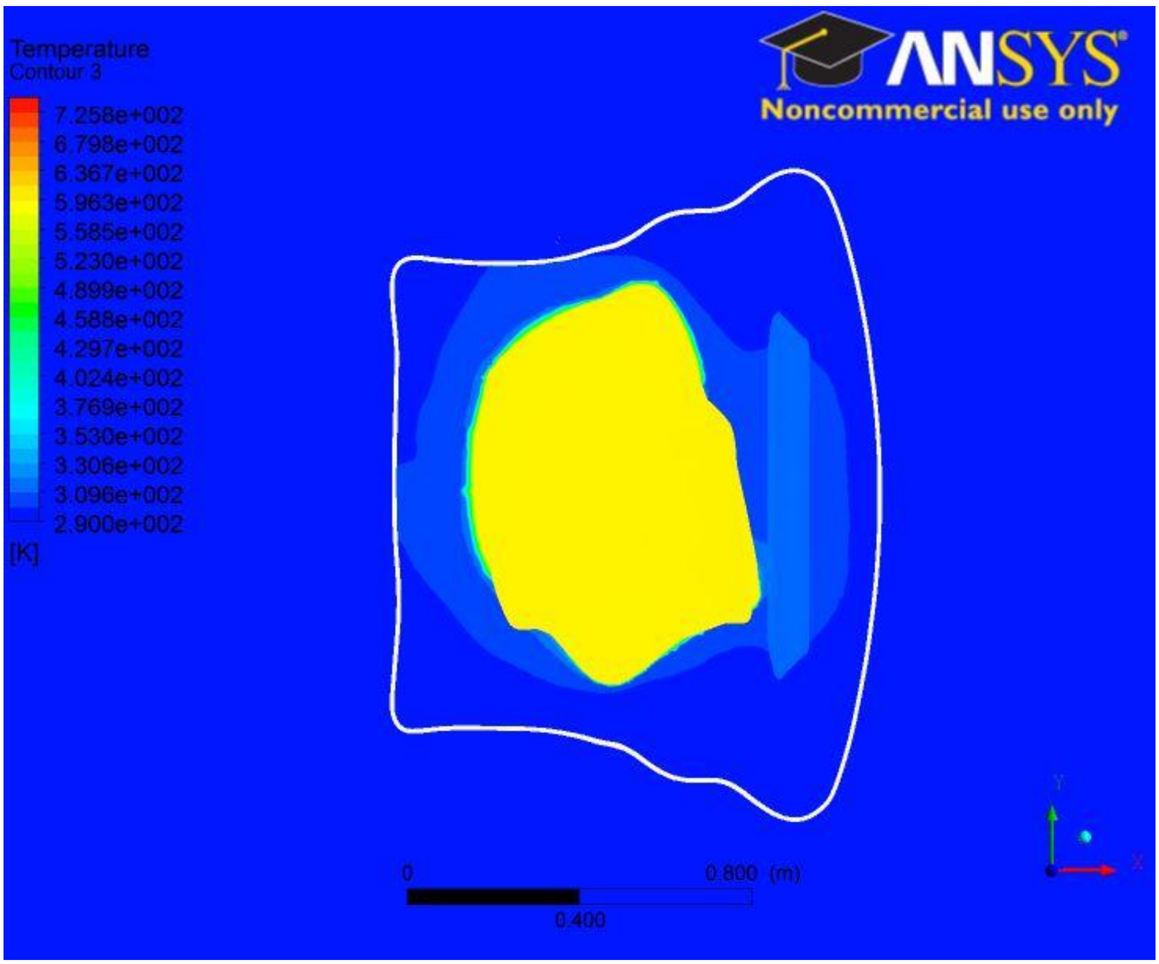


Figure 5.5: Temperature plot on a horizontal plane passing through the radiator and the engine

The temperature plot presented in Figure 5.5 ascertains the presence of high temperature zones in the engine vicinity. And the prime reason for this is the poor air circulation around the engine.

Also, the surface temperatures of all the components were obtained to assess the need for relocation of any component. In this section the surface temperatures obtained by the components for two different layouts as shown in the Figure 5.6 and 5.7 are presented.

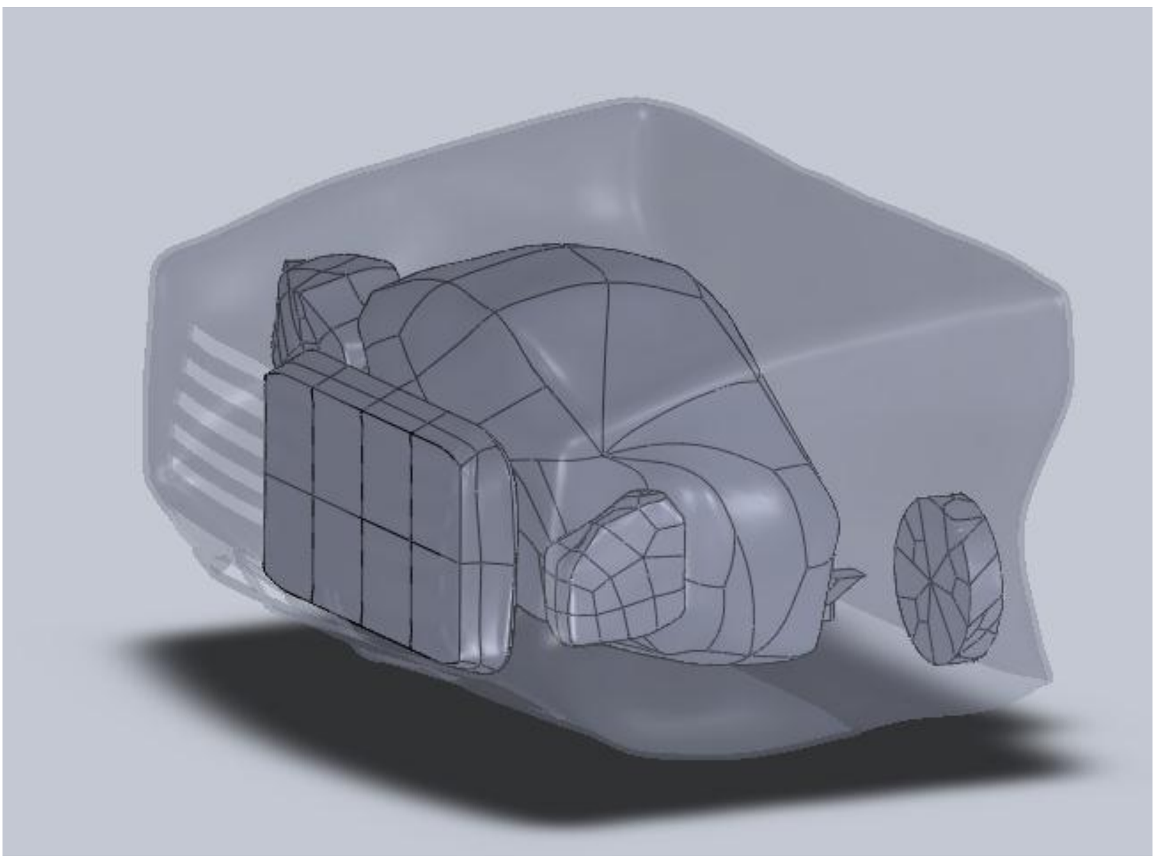


Figure 5.6: Initial Layout of the components in the car under hood with the radiator placed in front of the engine

For the initial layout of the components, under the car under hood, the radiator with constant heat rejection value, equal to 28000 watts, was placed in front of the engine as shown in Figure 5.6. The air flow path for this layout was defined through the grille inlet, radiator, and engine surface and then exited the compartment by flowing over the exhaust manifold surface. The maximum surface temperatures of the components thus obtained are tabulated and presented in the Table 5.1.

Table 5.1: Maximum surface temepratures of the components for the first layout

Component name	Maximum surface temperature obtained (K)
Radiator	330
Airfilter	298.18
Battery	315
Brake booster	299
Engine	600
Exhaust manifold	750
Coolant tank	298.15

The results indicate that the maximum surface temperature of the battery placed in the vicinity of the engine is above its threshold value, and thus relocation of this component should be considered. The inlet temperature of the auxilary fluid flowing through the radiator, placed in the path of the air flow, required for causing 28000 watts of fixed heat rejection, was found to be equal to 330 K for the initial layout.

The other layout of the components asessed in this project is shown below in the Figure 5.7. In this layout, the radiator is relocated form it's position in the front and is placed behind the engine.

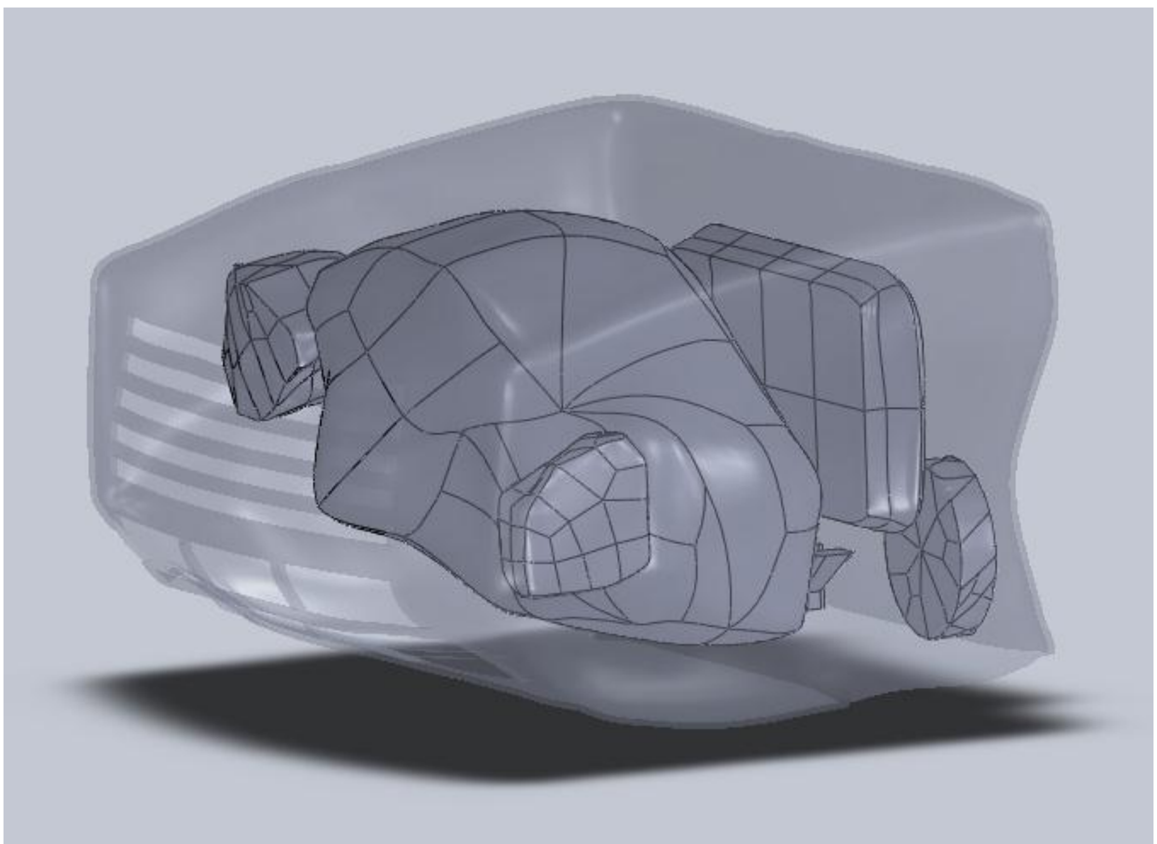


Figure 5.7: Layout of the components in the car under hood in which the radiator is placed behind the engine

The inlet temperature of the auxilary fluid flowing through the radiator, placed in the second layout as shown in the Figure 5.7, required for causing 28000 watts of fixed heat rejection, was way above its boil temperature. This suggested that, locating the radiator behind the engine proved to be hostile for the under hood thermal behavior and the intial layout should be preferred over the second layout of the components.

CHAPTER SIX

CONCLUSION

6.1 CONTRIBUTIONS

Implementation of a CFD simulation over the complex 3-Dimensional CAD geometry of a car under hood is presented in this thesis. Appropriate CAD clean up processes were implemented on the initial geometry and the geometry was made suitable for performing CFD simulatons upon it. A good quality mesh with 659538 nodes 3544757 elements was generated. The maximum skewness value of the mesh generated, was below 0.9 throughout, indicating that the good quality of the mesh obtained.

Porous medium model and the ungrouped macro heat exchanger model, were used to model the radiator. The use of these models significanlty simplified the modeling process. Thus, CFD simulations were carried out sucessfully on the complex under hood geomtery by implemeting the simplified models for radiator.The results thus obtained from the simulations performed indicated zones of thermal risk. The prime reason for the formation of these high thermal risk zones in the underhood was due to the imporper circulation of air through the tightly packed under hood space. The velocity vector plots obtained for the under hood geometry, indicated the presence of recirculation zones which further deteriorated the air flow circulation in the under hood space.

Also, in this project, CFD simulations were performed over two different layouts of the under hood components, to assess the need for relocation of the components in order to keep the surface temperatures of these components in a safe limit. The results thus obtained indicate that the placement of radiator behind the engine is detrimental for the under hood thermal behavior. Thus, the optimal placement of the components in the under hood space can significantly improve the under hood thermal behavior.

6.2 FUTURE WORK

Running the rigorous CFD simulations for preforming the optimization runs, to obtain the optimal placement of the components under the hood is highly time consuming and computationaly expensive. Hence, in future work, an approximate model using the neural networks or response surface methodologies can be built from the data obtained by performing few sample CFD simulations. Later, this approximate model can be used for carrying out the optimization runs.

REFERENCES

[1]. V B Gantovnik, S Tiwari, G M Fadel, and Yi Miao, “Multi-Objective Vehicle Layout Optimization”, 11th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference, AIAA-2006-6978, Portsmouth, Virginia, 2006.

[2]. S. Chacko, B. Shome, A.K. Agarwal and V. Kumar, “Underbody Simulation for Vehicle Thermal Management”, Altair CAE Users Conference,Bangalore,2006.

[3]. E. Weidmann, J. Wiedemann, T. Binner, and H. Reister, “Underhood Temperature Analysis in Case of Natural Convection”, SAE Technical Paper 2005-01-2045, 2005.

[4]. I.F. Hsu and W.S. Schwartz, “Simulation of the Thermal Environment Surrounding an Underbody Fuel Tank in a Passenger Vehicle Using Orthogonally Structured and Body-Fitted Unstructured CFD Codes in Series”, SAE Technical Paper, 950616, 1995.

[5]. N. Wenyan, “Lift Truck Underhood Cooling Simulation”, Counterbalanced Development Center, NACCO Materials Handling Group, 2006.

[6]. K. Srinivasan, G. Woronowycz and M. Zabat, “An Efficient Procedure for Vehicle Thermal Protection Development”, SAE Technical Paper 2005-01-1904, 2005.

[7]. T.G. Bancroft, S.M. Sapsford and D.J. Butler, “Underhood Airflow Prediction Using VECTIS Coupled to a 1-D System Model” In Proceedings of the 5th Ricardo Software International Users Conference, Shoreham-by-Sea, UK, 2000.

[8]. K.D. Huang and S.C. Tzeng, “Optimiztion of Size of Vehicle and Flow Domain for Underhood Airflow Simulaton”, Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering vol. 218 no. 9 945-951, 2004.

[9]. S. Chacko, B. Shome, and V. Kumar, “Numerical Simulation for Improving Radiator Efficiency by Air Flow Optimization”, Beta-Case International Conference, Greece, 2005.

[10]. D. Winnard, G. Venkateswaran and R. Barry, "Underhood Thermal Management by Controlling Air Flow," SAE Technical Paper 951013, 1995.

[11]. Ansys Workbench Design Modeler 13.0 Application Help.

[12]. SolidWorks Help Manual

[13]. Ansys 13.0 Help Manual

[14]. Ansys 13.0 Mesh Introduction

[15]. Application Briefs From Fluent, EX201, Underhood Thermal Management.

[16]. M. Khaledab, F. Harambatb and H. Peerhossainia, “Temperature and Heat Flux Behavior of Complex Flows in Car Underhood Compartment”, Applied Thermal Engineering Volume 30, pp. 590–598, 2010.

[17]. D. Jung and D.N. Assanis, “Numerical Modeling of Cross Flow Compact Heat Exchanger with Louvered Fins using Thermal Resistance Concept”. SAE Technical Paper, 2006-01-0726, 2006.

[18]. Ansys FLUENT 13.0 Theory Guide.

[19]. Ansys FLUENT 13.0 User Guide.

[20]. D.Allen, M.Lasecki, W.Hnatczuk and R.Chalgren, “Advanced Thermal Management for Military Application”, Proceedings for the Army Science Conference (24th), Florida, 2005.