<u>İSTANBUL TECHNICAL UNIVERSITY ★ INSTITUTE OF SCIENCE AND TECHNOLOGY</u>

OPTIMISATON OF A HEAVY DUTY DIESEL ENGINE COOLING SYSTEM USING COMPUTER AIDED FLUID DYNAMICS(FLUENT)

Master. Thesis by

Mech.Eng. Mehmet Levent Timur

Department : Mechanical Engineering
Programme: Automotive Engineering

Supervisor : Assoc. Prof. Dr. Doğan GÜNEŞ

MAY 2003

İSTANBUL TECHNICAL UNIVERSITY ★ INSTITUTE OF SCIENCE AND TECHNOLOGY
İSTANBUL TECHNICAL UNIVERSITY ★ INSTITUTE OF SCIENCE AND TECHNOLOGY

OPTIMISATON OF A HEAVY DUTY DIESEL ENGINE COOLING SYSTEM USING COMPUTER AIDED FLUID DYNAMICS(FLUENT)

Master. Thesis by

Mech.Eng. Mehmet Levent Timur

No: 503001504

Department : Mechanical Engineering
Programme: Automotive Engineering

Supervisor : Assoc. Prof. Dr. Doğan GÜNEŞ

MAY 2003

INDEX

ABBREVIATIONS

CFD	**:** Computer Aided Fluid Dynamics
2D	**:** Two dimensional
3D	**:** Three dimensional
V6	**:** Six cylinder V type engine
OM457	**:** The code of the motor used in Mercedes O403 bus
0403	**:** Model name of a Mercedes Bus
NOx	**:** Nitrogen oxides
Tet/Hybrid	**:** Type of mesh used in fluent

TABLE LIST

Page No

FIGURE LIST

SYMBOL LIST

$\mathbf{K_L}$	**:** Pressure Drop Coefficient
R	**:** Radius
V	**:** Velocity
$\mathbf{\Delta P}$	**:** Pressure Drop

OPTIMISATON OF A HEAVY DUTY DIESEL ENGINE COOLING SYSTEM USING COMPUTER AIDED FLUID DYNAMICS(FLUENT)

SUMMARY

The cooling system of a heavy duty diesel engine used in a Mercedes O-403 bus is examined in this study. Cooling system of heavy duty diesel engines can be examined in four circuits. The coolant circuit, oil circuit, intake charge air circuit and airflow circuit are the parts of the cooling system. These main circuits and their parts are explained in this study.

The air flow circuit is the most important part of cooling system for this study since the air flow through the cooling room is modeled and examined. The intercooler, radiator and fan are installed in the cooling room. And also there are parts like batteries and air filter that have resistant effect to air flow installed in the cooling room. After developing the geometrical and mathematical model for the cooling room this model is imported to fluent. The properties of the air flow through the cooling room is taken from fluent. By changing the installation angle of the fan in the geometrical model the resistant effects of the batteries and air filter to air flow decreased and as a result the mass of air passing through intercooler and radiator is increased. So the cooling capacity of the cooling system of a Mercedes O-403 bus is increased while using the same cooling system parts but changing their installation angles. This study is a very good example that shows the advantages of using CFD for design of a cooling system.

YÜKSEK GÜÇLÜ DİZEL MOTORLARI SOĞUTMA SİSTEMLERİNİN BİLGİSAYAR DESTEKLİ AKIŞKANLAR MEKANİĞİ (FLUENT) KULLANARAK OPTİMİZASYONU

ÖZET

Bu çalışmada Mercedes O403 otobüsleri'nin soğutma sistemi incelenmektedir. Yüksek güçlü dizel motorların soğutma sistemleri dört ana devre başlığında incelenebilir.Bunlar soğutucu devresi, yağlama devresi, motor hava devresi ve genel hava akış devresidir. Bu devreler ve bunlarda kullanılan parçalar çalışmada genel olarak anlatılmıştır.

Hava akış devresi tüm bu devreler içinde bu çalışma için en önemlisidir. Çalışmada soğutma odasındaki hava akışı incelenmektedir. Soğutma odasında soğutma sisteminin parçaları olan radyatör, hava radyatörü ve fan bulunmaktadır. Ayrıca bu odada hava akışına direnç etkisi yaratacak aküler ve hava filtresi de vardır. Soğutma odasının geometrik ve Matematik modeli hazırlandıktan sonra Fluent programına girilmiştir. Soğutma odasındaki hava akışına ait sonuçlar Fluent programından alınmıştır. Geometrik model üzerinde fan yerleştirme açısı değiştirildiğinde radyatör ve hava radyatörü üzerinden geçen hava miktarında artma dolayısıyla hava filtresinin ve akülerin hava akışına gösterdiği direnç de azalma sonucuna varılmıştır. Sonuç da aynı soğutma sistemi kullanarak sadece yerleştirme açılarında yapılan değişiklik ile O403 otobüslerinin soğutma sistemi kapasitesi arttırılmıştır. Bu çalışma tasarım aşamasında simülasyon kullanmanın avantajlarını ispatlayan bir örnek olarak gösterilebilir.

1. INRODUCTION

For many years, cooling system design was concerned only with providing sufficient cooling at maximum engine output conditions (low vehicle speed and high ambient temperature). This approach results in not achieving the best fuel economy and emission properties. The heat getting out the engine and the operating temperatures of the engine are directly proportional with the exhaust emissions. With demands of increased fuel economy, longer engine life, and reduced emissions, greater control of engine performance cannot be maintained when over cooling occurs.[1]

Ideally, engine cooling is undesirable from the thermodynamic point of view. If the heat transfer rates from metal could be reduce, then more power could be produced at a particular fuel flow rate. The other major and probably most important aspect of reducing heat transfer is that there is less heat to remove out of the radiator. However, in practice, due to metallurgical constrains the conventional engine metals can withstand only a certain maximum temperature level. At these higher temperatures, the fatigue stresses induced in the metal under cyclic load increase. Engine oil also losses its lubricating qualities when temperatures exceed (177 C) and, as a result, excessive engine wear occurs.[2] Thus, engine cooling is extremely necessary for good engine reliability and durability with a compromise of the thermal efficiency. An optimum system should offer the following advantages.[1]

1. Increase in coolant, oil, and engine metal temperatures during cold start or low ambient temperatures and light route load conditions.
2. Sufficient cooling of the oil and engine metal temperatures under uphill and full load conditions.

For the development of any technical product, one can advance in two ways. The physical processes within the product can be mathematically modeled to accurately define its performance and can also develop the product by laboratory test work.[3]

Each of these methods has their individual advantages. But with the advent of high-speed computers, which help in solving the problems, the former has the potential to be lower cost and less time consuming. In contrast to experimental approach, a simulation model helps the designer to analyze various configurations of the system.

Also, reducing the hardware prototype stages and making an increase in the use of computational design and simulation can achieve a major reduction in development time.[4] Engine cooling simulation tools have been used for many years. One-dimensional flow computation is the standard method for determining pressure losses and volumetric flow rates in automotive coolant circuits. For example, it allows the design of hoses to be supported during the development phase. Other applications of one-dimensional simulations are support for the design of coolant pumps and the computation of the heat emission of vehicle coolant circuits. The engine cooling system was also a pioneering application for three-dimensional flow calculation computational fluid dynamics (CFD). With the result that CFD computation is now state-of-the-art in the design of today crankcase and cylinder head coolant jackets. With the aid of these computations, good coolant flow is ensured for areas with high thermal stress around the combustion chamber and exhaust passages, with correspondingly low pressure losses in the cooling jacket.

In order to compute the temperature distribution in the components of the core engine, knowledge of the convective heat transfer coefficients is necessary.[3] In the finite element program, heat transfer coefficients computed by the CFD program are used and corrected locally with input from boiling models corresponding to the computed local component temperature. In an iterative process, the component temperature is taken from the CFD program in order to take account of the influence of the local coolant temperature on the heat transfer coefficient.

The aim of this thesis is to develop a model of a heavy-duty diesel engine cooling room. Mercedes O403 bus cooling room is examined in this study. A V6 engine (OM457) is used in 0403 buses.

In the cooling room of an O403 bus there are an air filter and batteries that have resistant effects to the airflow in cooling room. By developing a model of the cooling room, in this study the optimization of the installation locations of the main parts of the cooling system (radiator, intercooler) is the major object.

2. COOLING SYSTEM OVERWIEV

Cooling system can be examined in details basically in four main circuits.[1]

1. Coolant circuit comprising of the fuel pump, cab heater, oil cooler, engine, radiator/bypass thermostat, fan, fan actuator and radiator.
2. Oil circuit comprising the oil pump, oil filter, oil thermostat, oil cooler, regulator valve, oil gallery, main bearing circuit, crank bearing circuit, piston cooling circuit, accessories cooling circuit, and the oil sump.
3. Air Flow circuit comprising of the grill, inter-cooler, condenser, radiator shroud, fan and the branching in the compartment around the engine.
4. Intake air circuit comprising of the turbocharger, inter-cooler, engine and the exhaust.

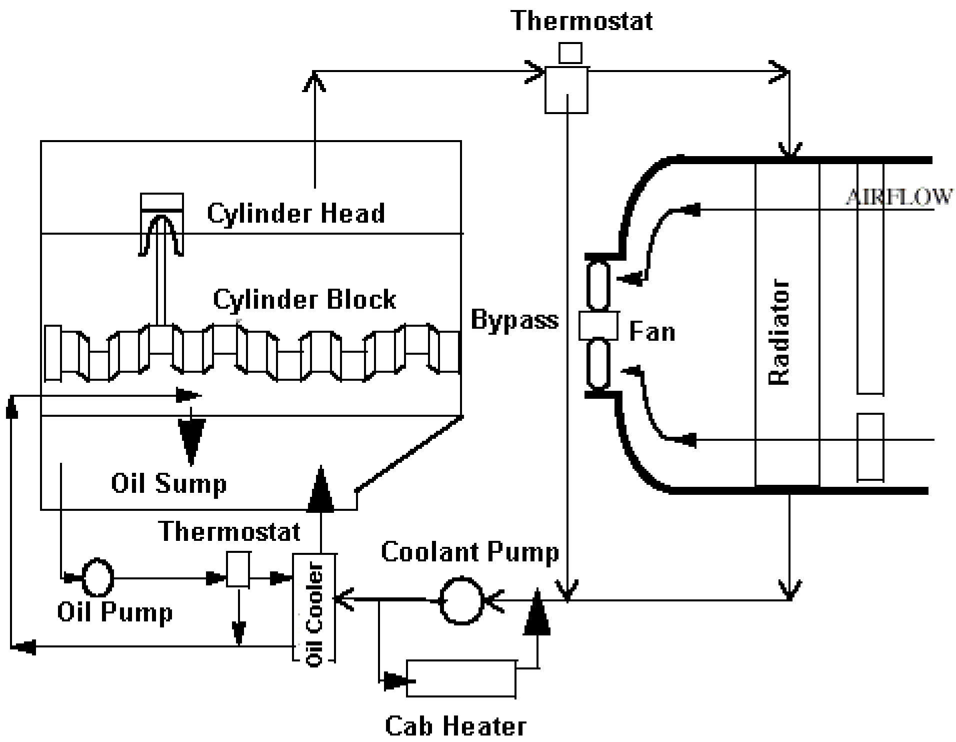

Figure 2.1. The Complete schematic of cooling system.

2.1. Coolant Circuit

The coolant system consists of the following major components: coolant pump, cab heater, engine, thermostat, fan actuator, and radiator. The pump is used to circulate the engine coolant. Pressurized coolant from the pump is forced through the oil cooler and the engine. Heat rejection from the engine is the main source of energy to the coolant. The thermostat is used to control the flow of coolant through the radiator, providing fast engine warm-up and regulating coolant temperature. When starting a cold engine or when coolant is below operating temperature, the closed thermostat directs all the coolant through the bypass to the pump. When the thermostat opening temperature is reached, coolant flow is divided between the radiator and the bypass tube.

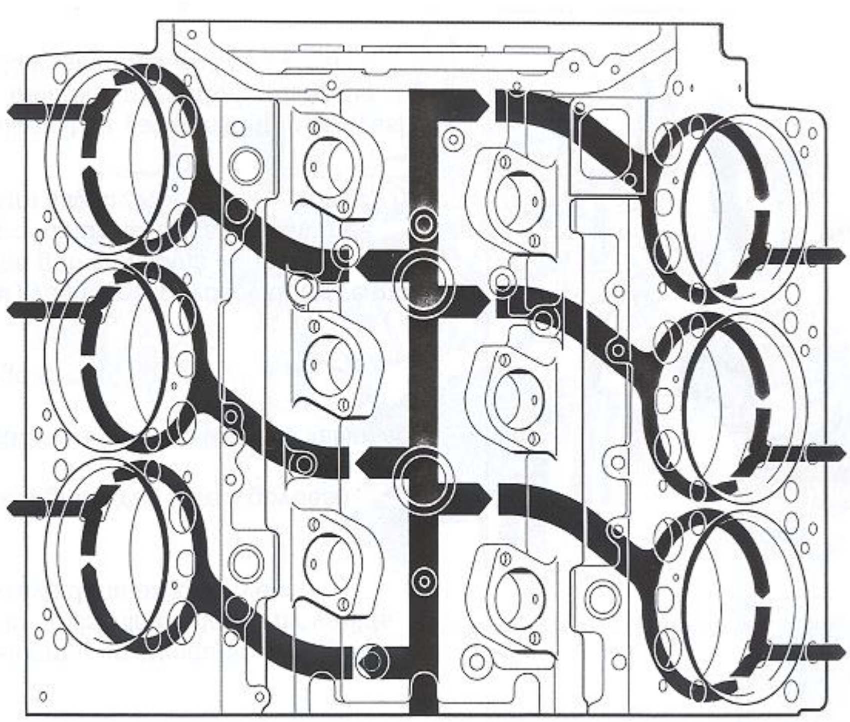

Figure 2.2. Coolant path of a heavy-duty Mercedes engine

2.1.1. Coolant Pump

The coolant pump circulates the coolant through the engine cooling system.[5] The pump can be driven directly off the engine by means of a gear mechanism or can be driven with a controlled electric motor. The power used by the pump can be calculated by standard pump laws. The coolant pump within the cooling system is assumed to have no effect on fluid temperature. The pumping of a fluid through the system generates an increase in thermal energy due to fluid friction (which is dependent upon fluid viscosity and system pressure). The most important parameter describing the pump is the rate at which the coolant is pumped.

2.1.2. Cab Heater

The cab heater unit in the coolant circuit removes energy from the system. The cab environment is not the main component of the cooling system, but the cab comfort and performance of the cooling system are coupled through the cab heater.

The cab environment control system consists of the cab heater unit with blower and insulated cab walls. Cab blower setting is dependent upon driver preferences. The desirable temperature inside the cab at all ambient temperatures is 21 to 27°C.[6] Hence, the blower settings are made to fluctuate accordingly to achieve this comfort temperature zone. After determining the flow rate through the blower, the inlet air temperature to the cab heater or the temperature out of the blower is determined based on the percentage of fresh air allowed to mix with re-circulated air. The inlet door position will allow for about 15 - 100% of fresh air operation. A certain percentage of fresh air is always admitted into the cab to reduce irritating smoke and dust effects. The inlet door may be controlled by a vacuum actuator or by gravity. The percentage of fresh air varies with vehicle speed for gravity-operated doors. As the vehicle speed increases, the amount of re-circulation air decreases.

The cab heater can be defined as a coolant to air finned heat exchanger. The heater is operational only when ambient temperatures are below specified cab comfort levels. To evaluate the average cab air temperature, it is necessary to know the heat energy which is absorbed by the cab walls and/or interior, the energy which is dissipated through the walls and/or the energy lost by infiltration of the ambient air (leakage of ambient air into the cab) in addition to the heat energy entering the cab from the heater.

2.1.3. Thermostat

Thermostats are temperature sensitive flow control valves. They are used to control the flow of coolant through the radiator and maintain the temperature within a specified range in the coolant circuit.[1] Their operation is in response to the temperatures sensed by the wax sensor. The thermostat sensor begins to open when the liquid into which it is immersed reaches the thermostat start-to-open temperature (also called the thermostat activation temperature), directing some liquid to flow along the new path. If the temperature continues to rise, the valve further opens until its full open temperature is reached. When this happens, maximum flow is directed through the radiator in the cooling system. As the system cools down, the reverse action takes place. The most common type of thermostat used is solid/liquid phase wax actuator. The brass housing (cup) in these kinds of thermostats is filled with a heat expansive wax material that is compounded to provide accurate, repeat-able temperature response. The wax is sealed within the cup by an electrometric sleeve. The sleeve envelops a polished stainless steel piston with tapered end. A seal and brass cover completes the sealing of the unit. The brass cup is in the path of the liquid flow. Heat is conducted through the walls of the cup to the wax. As the wax within the brass cup reaches its special compounded thermostat, it melts and expands, displacing the piston by squeezing the sleeve. The piston is in turn moved by the sleeve and the valve opens. As the temperature drops, the wax contracts allowing the piston to return. Operation of the plunger can be controlled mechanically or by varying the melting point temperature of the wax material (this can be achieved by adding varying amounts of copper filler). The different processes happening within the thermostat have to be carefully understood.

The different processes that take place in succession before the thermostat is actuated are: [1]

- Heat transfer by convection from the hot liquid to the outer surface of the brass cup (in contact with the coolant).

- Heat transfer by convection through the cup to its inner surface.

- Heat transfer by convection to and through the wax

- Melting and increase in volume of wax.

- Movement of the piston and finally the actuation of the valve.

2.1.4. Fan

Fan is the part of the cooling system that helps heat rejection process in the radiator and inter-cooler. Fan provides ram air circulation for the radiator and inter-cooler. For getting emission standards like EURO II the pressure gain of ram air from fan must be in some specified gaps.

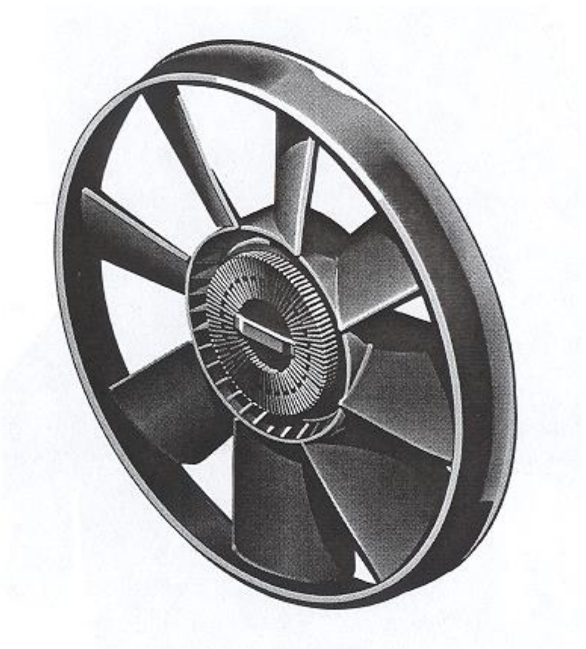

Figure 2.3 Fan used in a Heavy-Duty Mercedes Engine

Due to high-pressure drops in the radiator and intercooler, air back flow can occur after the fan. In the next figure you can see the design developments for preventing air back flow in cooling system of a heavy duty Mercedes engine.[5]

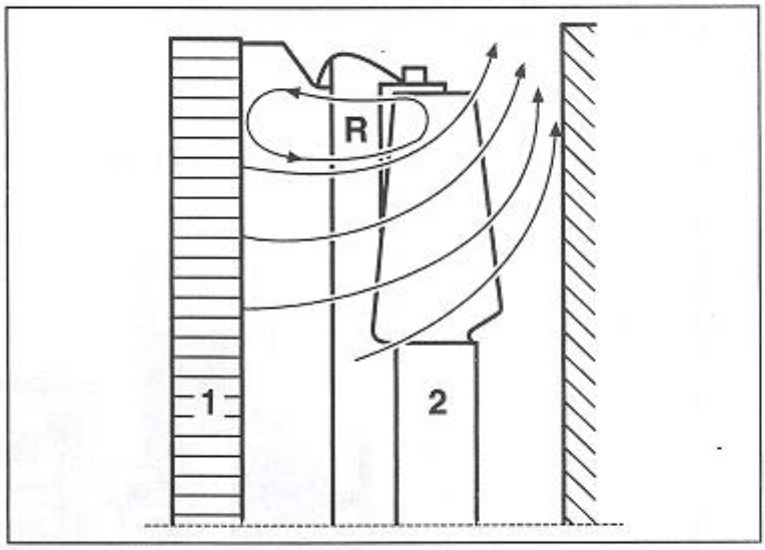

Figure 2.4.Air back flow in fan

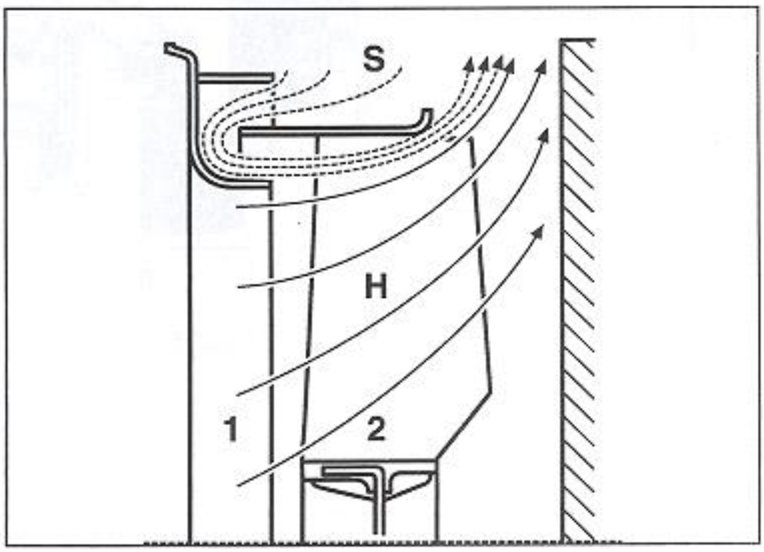

Figure 2.5.Design Improvements for preventing air back flow

2.1.5. Fan Actuator

Fan actuator activates the fan by controlling the coolant temperature and retarder temperature. In heavy-duty Mercedes diesel engines fan actuators works in two different positions. Below you can see the fan activation properties.[5]

Table 1 Fan Actuator Properties

	POSITION I		POSITION II	
	OPEN	CLOSED	OPEN	CLOSED
Coolant Temperature	T= 93 C	T= 90 C	T= 96 C	T= 93 C
Retarder Temperature	T=90,5 C	T=87 C	T= 94 C	T= 87 C

2.1.6. Radiator

The radiator is the most important heat exchanger in the engine cooling system as it transfers the majority of the thermal energy from the coolant to the surrounding air. The radiator is a liquid to air heat exchanger. It experiences a large variation in coolant inlet condition and flow rate. The radiator flow may be small or large or it may be extremely low and there will be no flow when the engine is shut down or the thermostat is completely closed. The flow depends both on the radiator thermostat position as well as the performance of the coolant pump. The operation of the thermostat is explained in the previous section. The coolant flows through the radiator, only when the coolant temperature reaches the thermostat control temperature. Hot coolant enters the radiator, where it is cooled by ram (ambient) air. Ram airflow is controlled by the shutters (if the system is so equipped) located at the back of the radiator and in front of the fan. Depending on the amount of cooling required, the shutters are open or closed. The coolant exiting the radiator then mixes with the coolant from the bypass and the coolant from the cab heater, and is then pumped to other parts of the coolant circuit.

Heat transfer in the radiator takes place by two modes, conduction and convection. Heat is transferred from the coolant to the wall by forced convection, by radial conduction from the inside of the wall to the outer surface, and from the outer surface of the tube to the ambient air by forced convection.

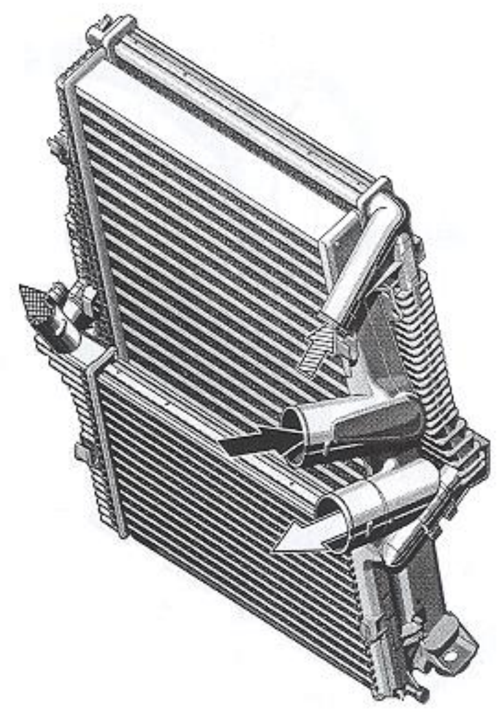

Figure 2.6. Radiator and inter-cooler combination of a heavy duty Mercedes engine

2.2. Oil Circuit

For analysis of the cooling system, a study of the oil circuit is extremely important as it has a direct effect on the coolant temperatures because the coolant gains energy from the oil in the oil cooler. Also, the oil system may be considered a cooling system in itself with oil as the cooling medium. The heavy-duty diesel engine oil circuit has two purposes. First of all, it is used to lubricate the engine parts to reduce engine wear and bearing load. Secondly, the oil serves as a coolant reducing the temperatures of the piston and bearings

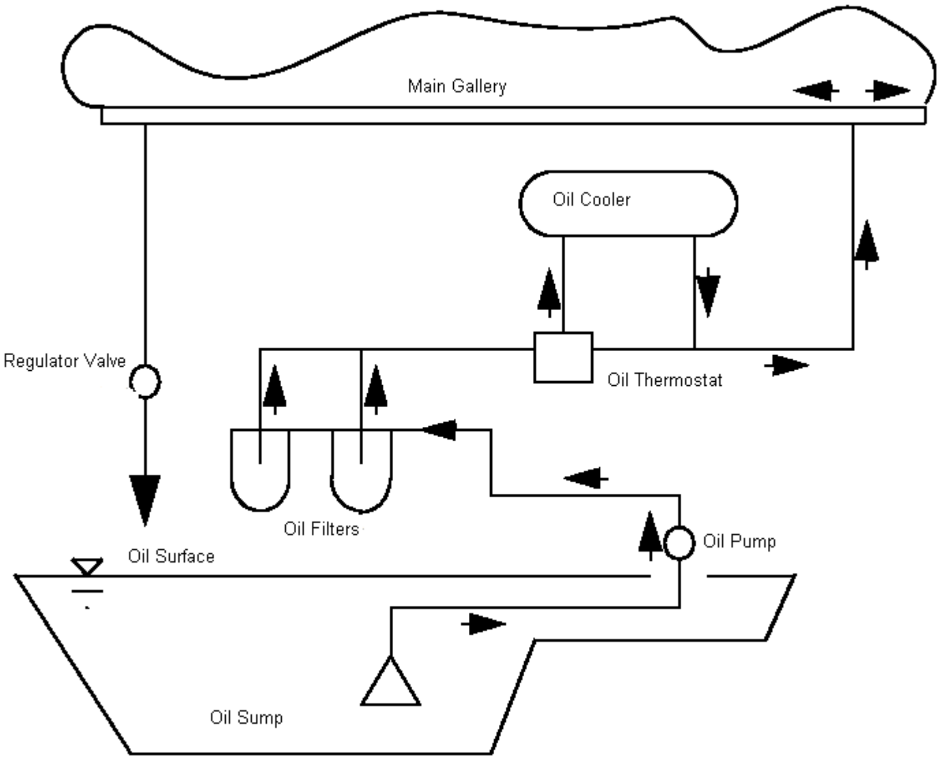

Figure 2.7. General oil circuit of a heavy-duty diesel engine

2.2.1. Oil Pump

The oil pump just like the coolant pump circulates the oil through the oil system. This pump is too driven by the engine. The oil pump contributes significantly to the heating of the oil. Hence, the viscous heating of the pump due to pump losses cannot be neglected. Rise in oil pressure and irreversible increase of the internal energy of the oil occurs at the pump due to pump input work. As the oil travels in the system, friction dissipation causes an increase in the internal energy of the oil. Essentially, the power used by the pump eventually turns into internal energy of the oil. Hence, the pump power can be used to calculate temperature rise in the oil pump.[5]

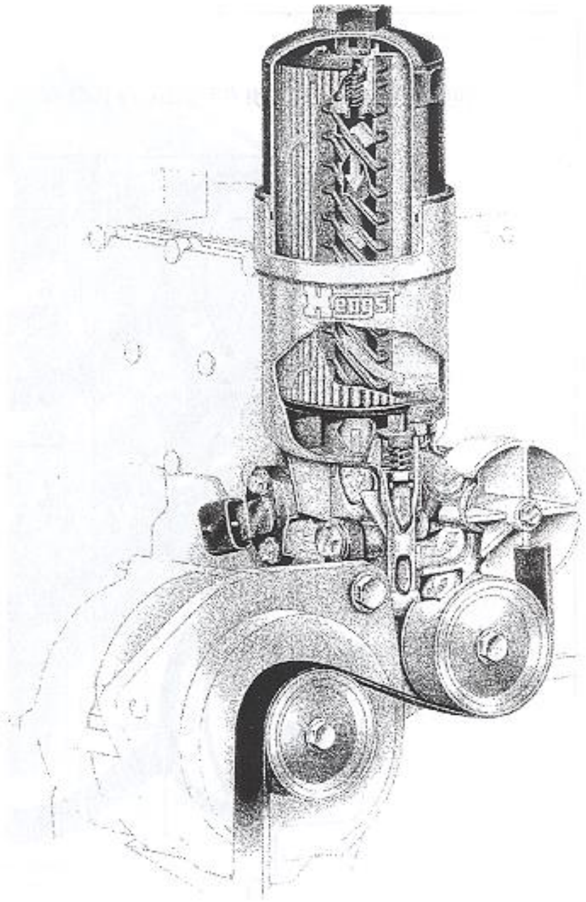

Figure 2.8. Oil pump of a Heavy Duty Diesel Engine

2.2.2. Oil Filter

The oil filter is considered to be an additional oil sump because of its size in this system. Incoming oil mixes with that contained within the filter to determine a representative oil filter temperature. The effect of oil filter ambient heat loss is considered negligible.

2.2.3. Oil Thermostat

The oil thermostat works in the same logic as the coolant thermostat. After reaching to the previously set activation temperature, the oil thermostat fully gets open and sends the whole oil to the oil cooler.

2.2.4. Oil Cooler

Oil cooler is a heat exchanger that is used to transfer the heat of oil to the coolant. The pump oil from the sump passes through the filters to the oil cooler. During the warm up period, a small amount of oil flows. During the working of the thermostat, the oil flowing through the oil cooler transfers heat to the coolant and the cooled oil is directed towards the regulator valve.

2.2.5. Heat Transfer In The Oil Circuit

There are basically three major sources of oil heating oil pump internal energy increase, bearing internal energy increase, and the piston cooling heat addition.

Bearing flow can be determined based on two components:[1]

1. Flow associated with the bearing hydrodynamic pumping action.
2. Flow due to external pressure feed.

The external pressure feed is necessary to effectively and efficiently remove energy from the bearings. This pressure generally provided by the main rifle. The main rifle pressure can be found using experimental maps of rifle pressure versus engine speed. Under normal operating conditions, the rifle pressure is maintained relatively constant by the pressure regulator.

2.2.6. Oil Sump

At all operating conditions, about 50% of the oil in the system is contained in the oil sump. Hence, the oil sump temperature is the measure of the performance of the oil system. All the energy rejection from the main and big end connecting rod bearings, the piston cooling and the oil pump determines the oil sump oil temperature.

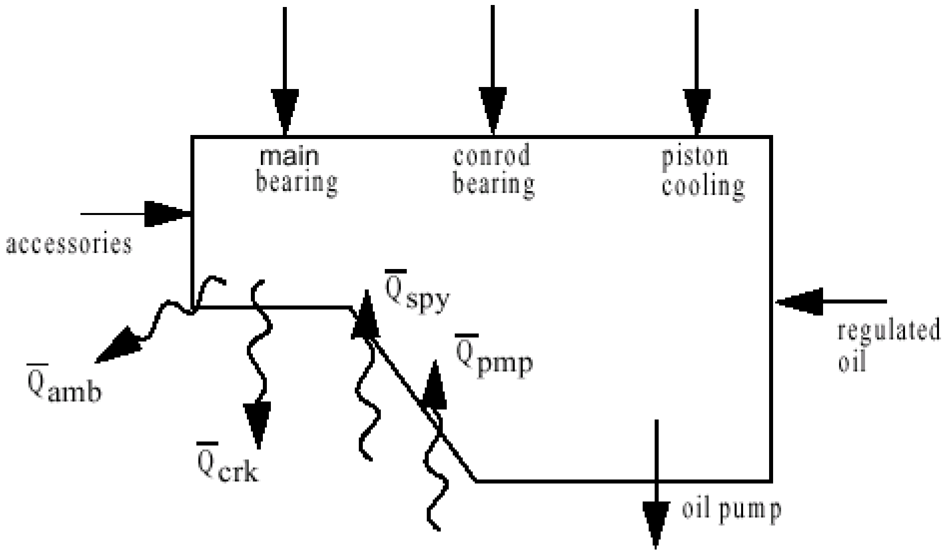

Figure 2.9. Thermodynamic model of oil circuit.

The first law of thermodynamics can be applied to the oil sump control volume in order to obtain an energy balance.

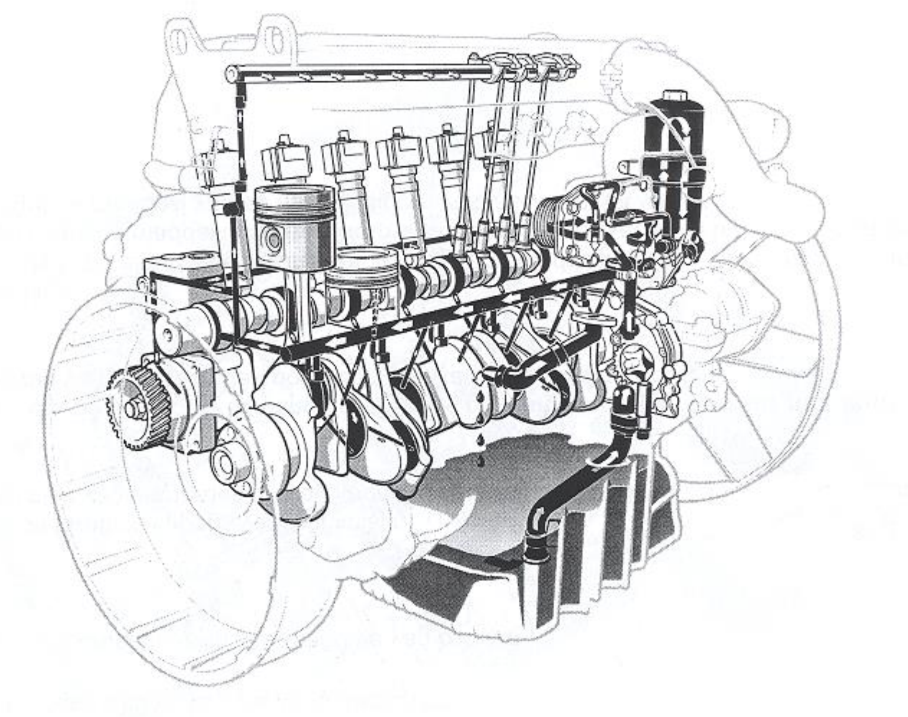

Figure 2.10. Oil Circuit of Mercedes Engine

2.3. Intake Charge Air Circuit

It consists of the turbocharger, inter-cooler and transient pipe lines connecting turbocharger inter-cooler and inter-cooler and the engine. After the combustion process and during the exhaust stroke, hot gases flow into the exhaust manifold. The exhaust gases have temperatures ranging from 400 to 700°C [6], depending on the speed and load of the engine. Then the gases flow to the turbine of the turbocharger. The turbocharger uses this energy to compress the air going into the engine. In the process, the temperature of the intake air increases. The inter-cooler reduces the air temperature before entering the engine for combustion.

2.3.1. Turbocharger

The turbocharger is made of two primary elements: a turbine and a compressor. It is one of the essential elements in a high specific power diesel engines. The primary role of the turbocharger is to use the energy from the exhaust gases to provide a higher air mass flow rate into the engine. This provides a higher mass of air in the cylinder and at the same time allows a proportionally greater amount of fuel to be added during each cycle. The additional fueling ability, for a given engine, will increase the maximum power output capability of the engine. Typically, as the high temperature exhaust gases flow to the turbine, part of the energy in the gas is converted to output shaft work. The mechanical energy through the shaft is used to drive the compressor. Hence, the compressor uses the work to compress the air to higher pressure and temperature (greater density).

2.3.2) Inter-cooler

The air coming out of the turbocharger is at an increased temperature. To increase the intake air density into the engine, the air is cooled in the inter-cooler. Hence, the inter-cooler is placed between the turbocharger and the engine. This results in increased air mass to the engine, making it possible to burn more fuel and make the combustion more complete. The combustion starts at a lower temperature thus reducing peak temperatures and pressures in the engine. Also, the inter cooling provides other improvements such as fuel economy, increased power density and reduced emissions, particularly particulate matter and oxides of nitrogen. All of these factors result in reduced thermal and mechanical stresses compared to the non-aftercooled turbocharged engine resulting in increased durability and reliability.

The inter-coolers can be of two types, air-to-water and air-to-air type. Previously in the low flow system; the inter-cooler or the aftercooler used was an air to water type heat exchanger. It was positioned within the coolant circuit after the radiator. The inlet air in this case could be cooled to a certain desired degree during light load conditions and at low ambient temperatures, thus improving the combustion. However, the coolant being at higher temperature than the ambient air, the degree of cooling that can be achieved at high loads is limited. The other one is air-to-air cooler. A lower engine intake air temperature is obtained by directly utilizing the ambient air as the cooling medium. By providing this kind of a cooler, the temperature can be reduced to about 22-28°C [6] higher than the ambient temperature. However, the air-to-air system while achieving a higher degree of cooling, introduces more resistance to be overcome by the fan in the ram airflow. Hence, higher temperature and less air flow over the radiator.

2.3.3. Inter-cooler Pipes

Air pipelines serve as connecting conduit between the turbocharger and the inter-cooler and from the inter-cooler to the engine intake manifold. During passage of the air through the pipelines, there is a drop in the temperature and pressure of the charge air. This is because of the heat loss to the surrounding ambient air, which is at a lower temperature than the charge air and the

2.4) Airflow Circuit

The coolant and the oil cooling system are designed to limit the temperatures in the various engine parts. Therefore, how to design the cooling system for sufficient cooling airflow during worst-case situations with maximum power consumption has always been a major concern. The worst-case situations are high ambient temperature, full load, and low vehicle speed, such as the case encountered in going up a steep mountain in desert conditions.[1]

Cooling airflow is either generated due to the ram air effect of the vehicle motion and/or by the cooling air fan. Because of the complex geometry, complex airflow patterns are induced along the flow passage through the intercooler, radiator, shroud, and over the engine. These flows are not easily understood even from good experimental studies.[7] To achieve a more precise prediction of the engine cooling system performance, an analytical model of the cooling airflow system must be developed.

The airflow through the radiator and the intercooler is directly proportional for rejecting process of heat from the coolant and engine air: The more airflow through these parts and the most effective cooling. So it is a very important fact to supply efficient airflow through radiator and intercooler while designing a cooling system. The importance of making an effective air flow circuit is higher in buses than cars. Because supplying enough airflow for a car cooling system is easier because of the fact that the engine and radiator are in front of the car where a direct contact of air occurs. But in an O403 like many of the other buses the engine and the cooling system are at the back part of the vehicle.

Since the aim of this study is to examine the resistant effects of the cooling system parts in the cooling room, airflow circuit is the most important one. By developing 3D model of the cooling room the airflow in the cooling room and the resistant effects of the cooling parts for different installation angles are calculated.

3. ENGINE ANALYSIS

The properties of the Mercedes Heavy Duty diesel engine that investigated are stated at the table.

Table 3.1 Mercedes OM 457 Diesel Engine Properties

Number of Cylinders	6
Max (RPM)	2000
Max Power	300 hp
Max Torque	1250 Nm @ 1400RPM
Compression Ratio	17.25: 1
Cylinder Bore	128
Stroke	155
Volume	12 lt
Mean Piston Speed	10.8 m/s @ 2000 RPM
Mean Eff. Pressure	16.1 Bar @ 2000RPM
Total Engine Weight	900 kg
Rotation (fly wheel side)	C.C.W
Cooling Water Capacity (Without re-cooling system)	16 lt
Max Lubrication Oil Capacity	30 Lt
Air Consumption	30.3 m3/min @2000 RPM

Figure 3.1.Mercedes OM 457 Diesel Engine

3.1. Heat Transfer Of The Engine

Heat transfer in the engine affects its performance, efficiency, and emissions. If the heat transfer within the engine is reduced, problems such as thermal stresses in the high temperature regions, and deterioration of the lubricating oil film can occur. An increase of heat transfer to the combustion chamber walls will lower the gas temperature and pressure within the cylinder, and in turn will reduce the work per cycle transferred to the piston. Hence, heat transfer between the gas in the cylinder and the gas-exposed cylinder surfaces is important from three aspects:[1]

1. The effect on the thermodynamic cycle and its efficiency and emissions,

2. The effect of the resulting metal temperatures on the operation of the different components, and

3. The energy transferred to the coolant and the oil in turn needs to be removed from the system by the radiator.

Changes in the gas temperature due to heat transfer impact pollutant emission formation processes. Generally, a higher temperature in the cylinder during combustion ill cause an increase in the NO x emissions, but a lower temperature in the cylinder will affect the oxidation f the particulate matter. Therefore, in the sense of emissions control, attention should be focused on engine heat transfer.

To analyze heat transfer of an engine, it must be divided into different control volumes.

3.1.1. Cylinder Block

The cylinder block can be divided into two control volumes: the cylinder liner surface and the cylinder block mass. The liner surface control volume receives heat from the in-cylinder gas and piston and rejects it to the block mass control volume. The volume that receives heat from the liner surface control volume rejects heat to the block coolant and to the engine compartment air.

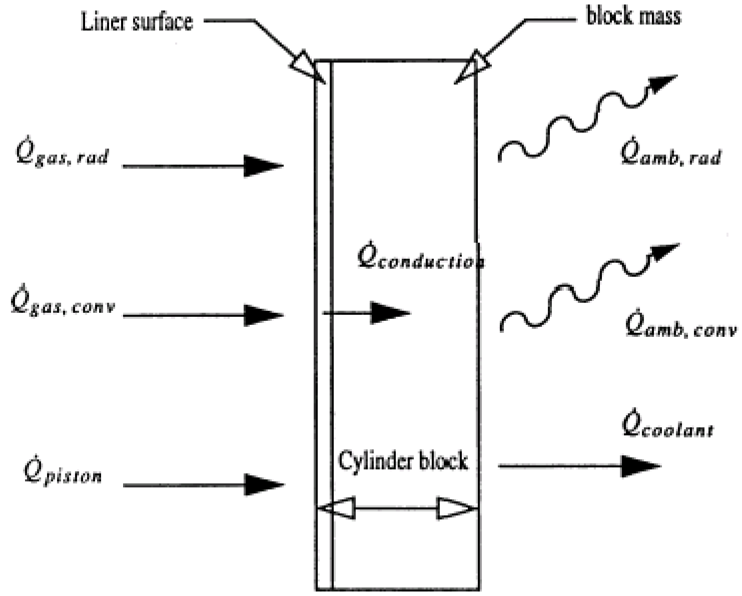

Figure 3.2 Engine cylinder block control volume

3.1.2. Cylinder Head

Similar to the cylinder block, the cylinder head can be also divided into two control volumes: the head surface and the head mass. The head surface volume receives heat from the in-cylinder gas and exhaust gases, and rejects heat to the cylinder mass. Whereas, the cylinder mass reject heat to the coolant, and engine compartment air.

3.1.3. Piston

Similar analysis can be performed for the piston surface and the piston mass control volume. The piston rejects heat to the cooling oil.

3.1.4. Piston Cooling Oil

The piston cooling oil control volume receives heat from the piston under crown

4. CFD (FLUENT) ANALAYSIS OF THE COOLING ROOM

To develop a successful model, first of all a geometric model of the cooling room is designed. And then the properties of the cooling parts like pressure drop, etc… are examined in the mathematical model. Both in geometrical model and mathematical model some assumptions are done. The reasons of these assumptions are explained.

4.1. Geometrical Model

The photographs of real O403 buses are as follows,

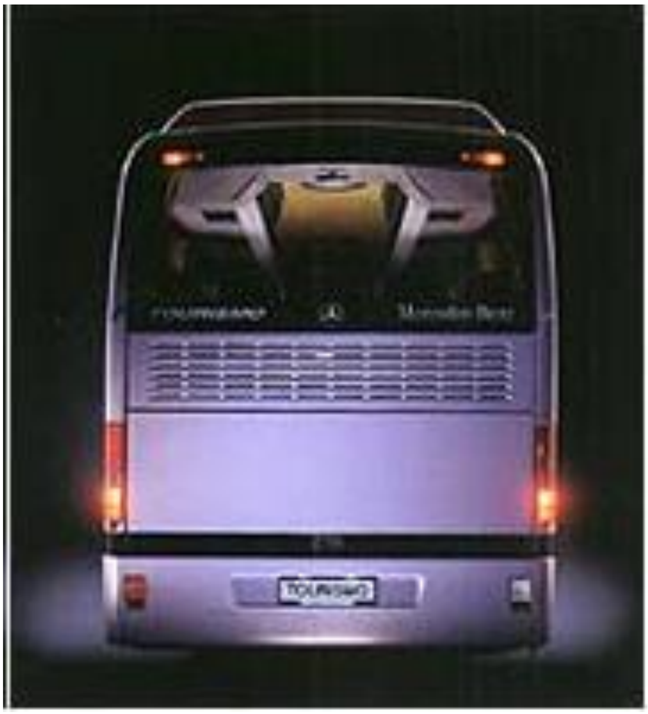

Figure 4.1 Back Side Photo of O403

Figure 4.2 Left Side Photo of O403

In the next figures you can see the technical drawings for the cooling room of O403 bus.

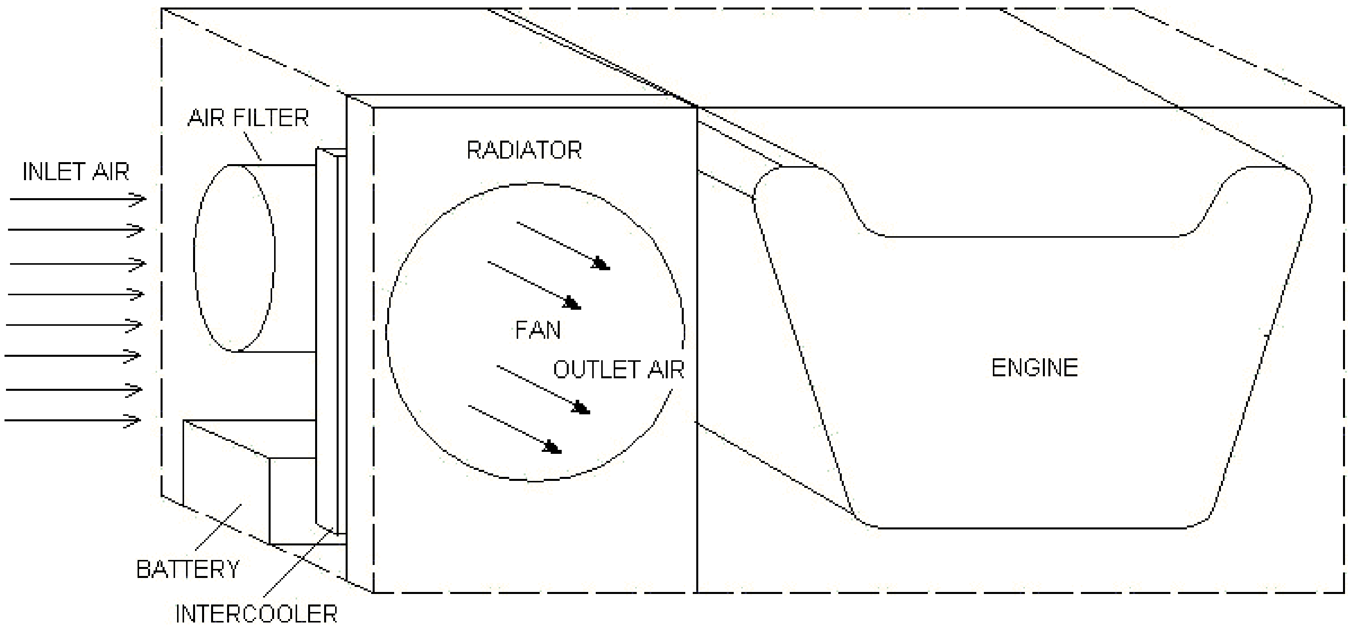

Figure 4.3 The 3D model of the cooling room

The air enters the cooling room from mass flow inlet face. It flows through the inter-cooler and radiator, and exits the cooling room after passing fan.

The batteries and air filter in the cooling room have resistant effects to air flow. Also there are another parts that have possible resistant effects in the cooling room, like inter-cooler pipes, radiator installation parts, air filter pipe, etc…are neglected in this geometrical model.

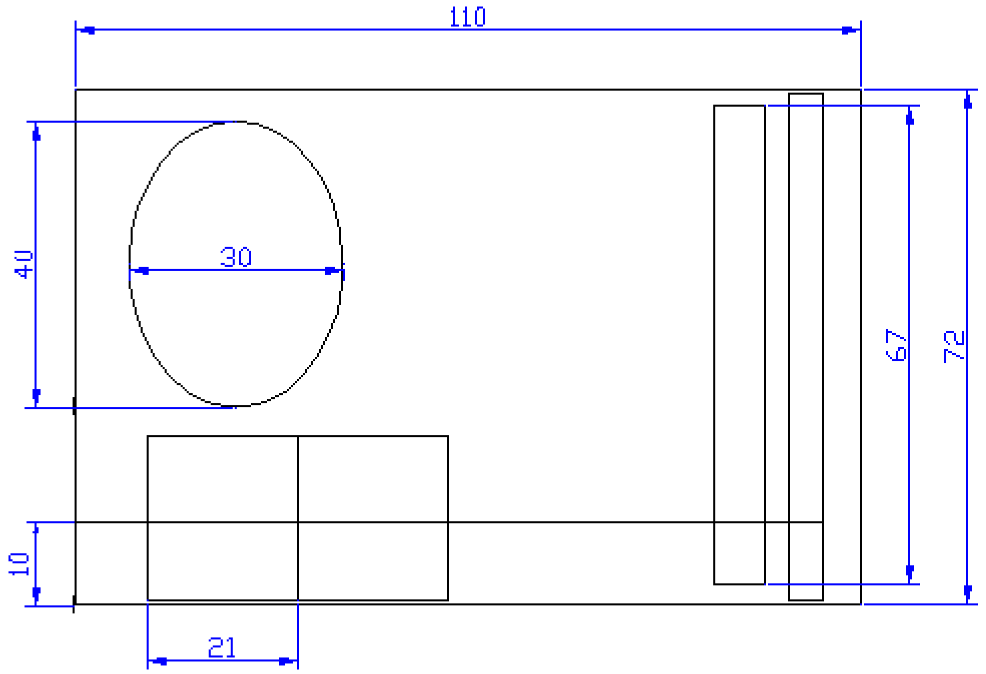

Figure 4.4 Front view of the cooling room

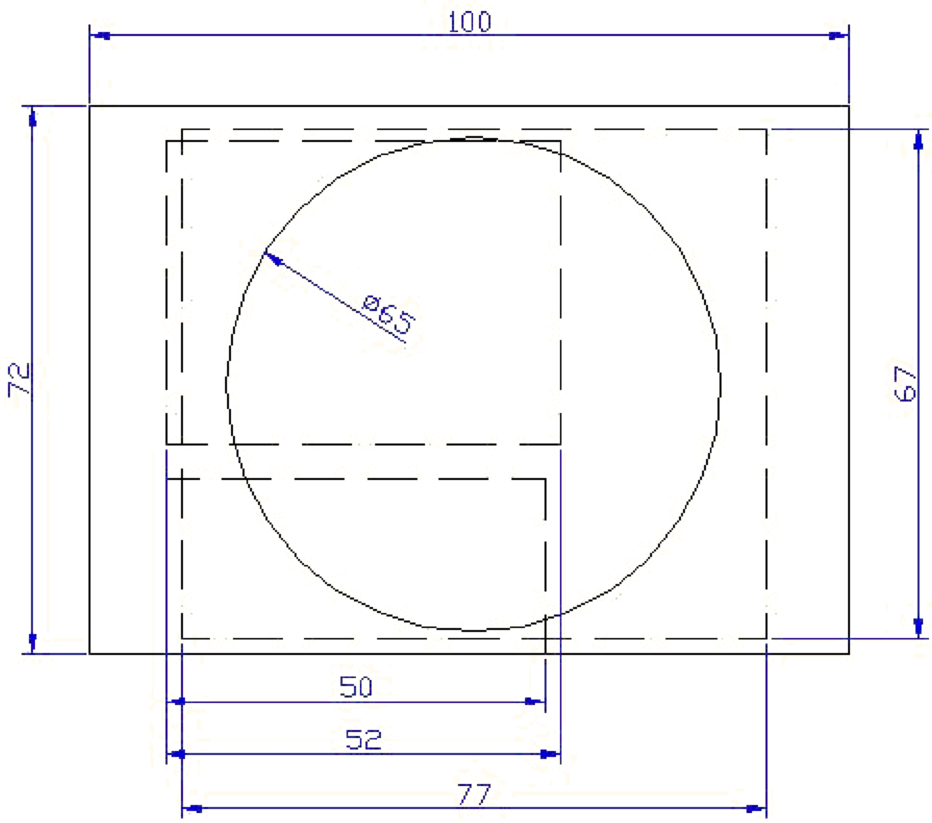

Figure 4.5 Side View of the cooling room

The following assumptions were done while developing the geometrical model.

1. The geometry of mass flow inlet face is ignored.

In real cooling room of a Mercedes O403 bus, there is a cover that can make resistant effects to air flow through cooling room. Because the main role of this study is to examine the resistant effects of the main parts in cooling room (air filter, battery) the geometry of this cover is not imported to the model.

2. The geometry outside the cooling room and its possible effects to air flow is ignored

3. The cooling pipes between the engine and cooling room are neglected

4. The geometry of fan shroud is neglected.

5. The geometry of air filter is accepted as a sphere.

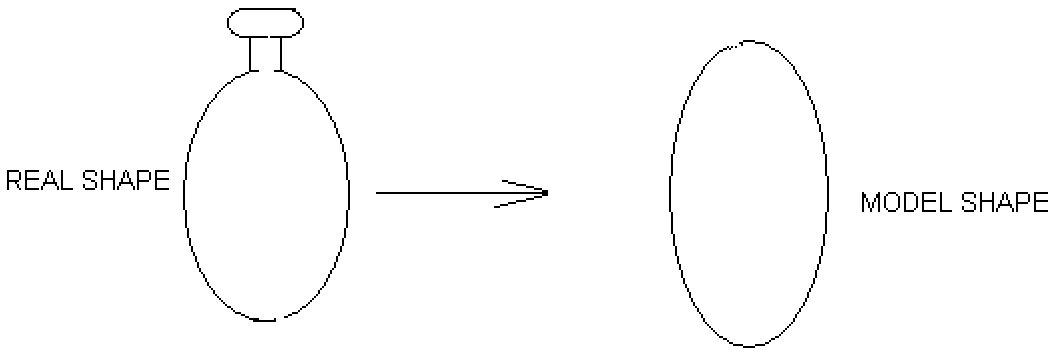

Figure 4.6 – The air filter geometry assumption

6. The width of radiator, inter-cooler and fan are neglected and they are accepted as 2d faces. And also to make a successful mash the dimensions are accepted as follows,

Table 4.1 Dimensions of Radiator and Inter-cooler

	Real Dimensions(mm)	Model Dimensions(mm)
Radiator	1000x711x49	1000x720
Inter-cooler	775x675x73	780x680

After making these assumptions, the geometrical model of the cooling room is designed in GAMBIT. Tet/Hybrid mash property of GAMBIT is used with a spacing of four in geometrical model, and the result is as follows,

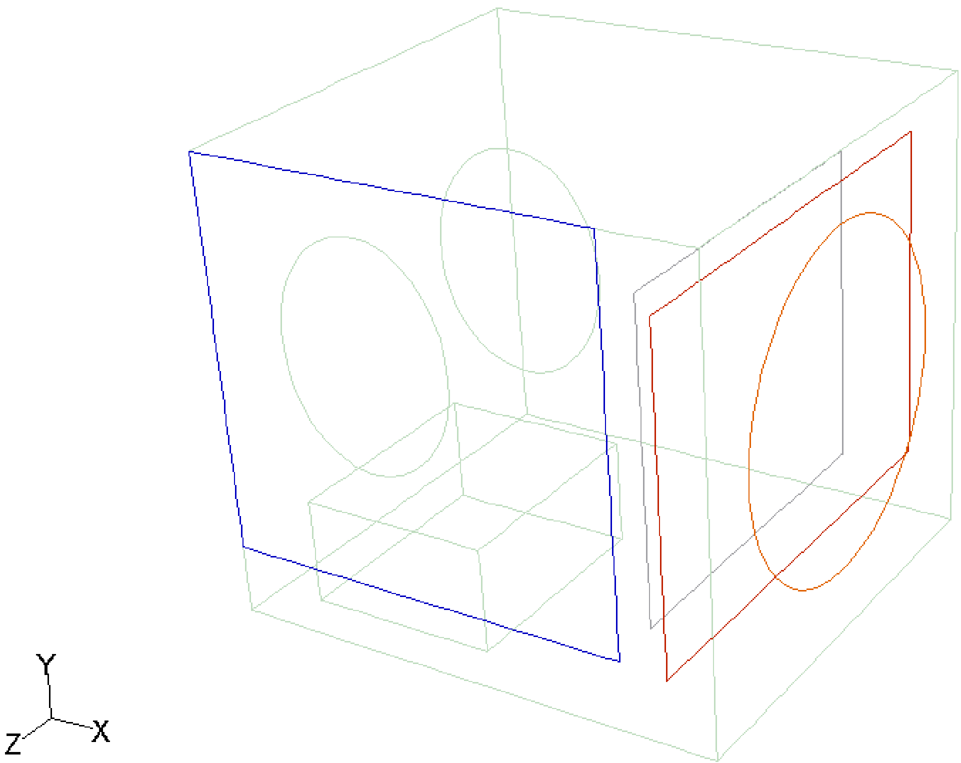

Figure 4.7 Outline of the Geometrical Model

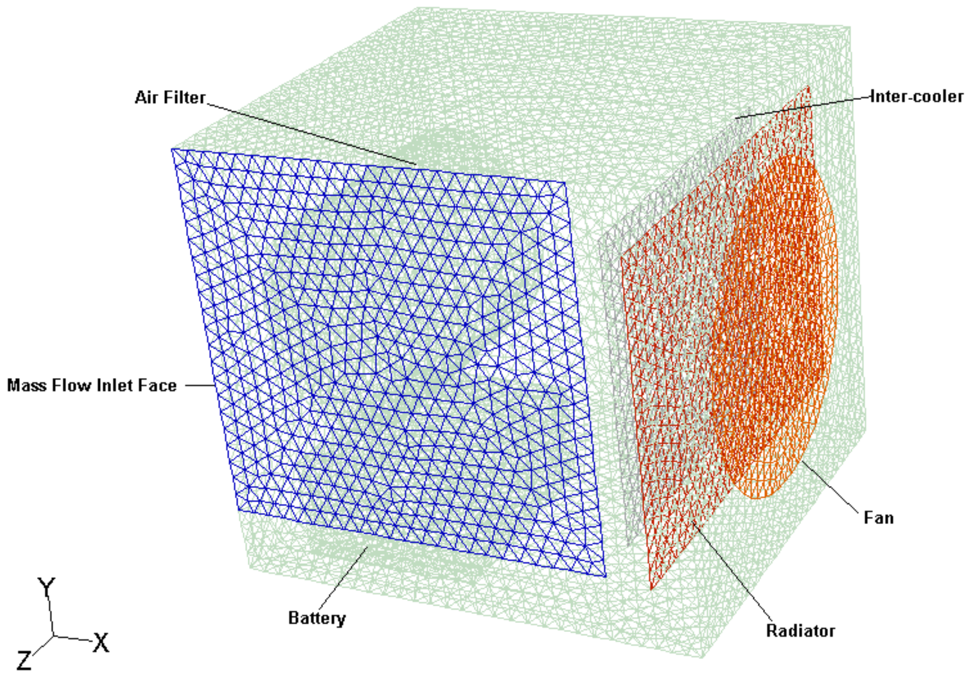

Figure 4.8 Geometric model developed in GAMBIT

4.2. Mathematical Model

Fluent models fluid flow by solving a set of conservation equations for mass, momentum, and energy.

4.2.1. Navier-Stokes Equations

For laminar flows, fluent solves the Navier-Stokes equations. The Navier –Stokes equations are based upon three universal conservation laws which are applicaple to fluid flows:

- Conservation of mass
- Conservation of momentum (Newton's Second Law)
- Conservation of energy

These equations are discussed in detail below.

The Mass Conservation Equation:

The equation for conservation of mass, or continuity equation , can be written as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial xi}(\rho ui) = Sm \qquad (4,1)$$

Equation 4,1 is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows. The source S_m is the mass added from any user defined sources.

For 2D axisymmetric geometries, the continuity equation is given by

$$\frac{\partial p}{\partial t}+\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial r}(\rho v)+\frac{\rho v}{r}=Sm \qquad (4,2)$$

where x is the axial coordinate , r is the radial coordinate, u is the axial velocity and v is the radial velocity.

Momentum Conservation Equations

Conservation of momentum in the I'th direction in an inertial (non accelerating) reference frame is described by,

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial p}{\partial x_i}+\frac{\partial \tau_{ij}}{\partial x_j}+\rho g_i+F_i \qquad (4,3)$$

where p is the static pressure, T_{ij} is the stress tensor (described below), and ρg_i and F_i are the gravitational body force and external body forces. (e.g., that arise from interaction with the dispersed face) in the I direction, respectively. F_i also contains model dependent source terms as porous-media and user-defined sources.

The stress tensor T_{ij} is given by,

$$\tau_{ij}=\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]-\frac{2}{3}\mu\frac{\partial u_l}{\partial x_l}\delta_{ij} \qquad (4,4)$$

where μ is the molecular viscosity and the second term on the right hand side is the effect of the volume dilation.

For 2D axissymetric geometries, the axial and radial momentum conservation equations are given by,

$$\frac{\partial}{\partial t}(\rho u)+\frac{1}{r}\frac{\partial}{\partial x}(r\rho uu)+\frac{1}{r}\frac{\partial}{\partial r}(r\rho vu)=$$

$$-\frac{\partial p}{\partial x}+\frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(2\frac{\partial u}{\partial x}-\frac{2}{3}\left(\nabla.\vec{v}\right)\right)\right]+\frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(\frac{\partial u}{\partial r}+\frac{\partial v}{\partial x}\right)\right]+F_x \qquad (4,5)$$

and,

$$\frac{\partial}{\partial t}(\rho v)+\frac{1}{r}\frac{\partial}{\partial x}(r\rho uv)+\frac{1}{r}\frac{\partial}{\partial r}(r\rho vv)= \qquad (4,6)$$

$$-\frac{\partial p}{\partial r}+\frac{1}{r}\frac{\partial}{\partial_x}\left[r\mu\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial r}\right)\right]+\frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(2\frac{\partial v}{\partial r}-\frac{2}{3}\left(\nabla.\vec{v}\right)\right)\right]-2\mu\frac{v}{r^2}+\frac{2\mu}{3r}\left(\nabla.\vec{v}\right)+F_r$$

Where,

$$\nabla.\vec{v};=\frac{\partial u}{\partial x}+\frac{\partial v}{\partial r}+\frac{v}{r} \qquad (4,7)$$

Energy Conservation Equation

Conservation of energy is described by,

$$\frac{\partial}{\partial t}(\rho E)+\frac{\partial}{\partial x_i}\left(u_i(\rho E+p)\right)=\frac{\partial}{\partial x_i}\left(k\frac{\partial T}{\partial x_i}-\sum h_j J_j+u_j\tau_{ij}\right)+S_h \qquad (4,8)$$

Where E is the total energy per unit mass, k is the conductivity, J_j is the diffusion flux of species $j^{'}$, and source term S_h includes heat of chemical reaction, any interphase exchange of heat sources defined.

The total energy E is defined by

$$E=h-\frac{p}{\rho}+\frac{u_{i^2}}{2} \qquad (4,9)$$

where the sensible enthalpy h is defined for ideal gases as

$$h=\sum_{J'} m_j^{'} h_{j'} \qquad (4,10)$$

and for incompressible flows as

$$h = \sum_{j'} m_{j'} h_{j'} + \frac{p}{\rho} \tag{4,11}$$

In equations 4,10 and 4,11, $m_{j'}$ is the mass fraction of species j' and

$$h_{j'} = \int_{T_{ref}}^{T} c_{p,j'} dT \tag{4,12}$$

Where T_{ref} is 298.15

4.2.2. Mathematical Model Of The Radiator

The pressure drop and jump property for the parts radiator, inter-cooler and fan are also examined in the mathematical modeling of the cooling room.

The radiator and inter-cooler are heat exchangers. Radiator transmits the heat energy of coolant to the air passing through. The amount of heat energy that can be rejected by radiator is directly proportional to mass of air flowing through it. While air flows through the tubes of radiator besides getting the thermal energy the pressure of the air passing reduces. The situation in inter-cooler is same.

The amount of pressure drop occurring in the radiator, and inter-cooler changes with the velocity of air passing through. You can see the experiment results of pressure drops for different velocities in table

Table 4.2 Experimental results for Radiator.

V (m/s)	ΔP (Pascal)
1,76	62
3,78	286
4,43	392
5,43	590

To get the most accurate result of pressure drops occurring the best way is finding the curve that fits best to this experimental results and then calculate the equation for this curve. But if you import an equation as a boundary condition for radiator the iterating time of fluent increases a lot. Because the main object of this study is to examine the resistant effects of batteries and air filter in cooling room instead of calculating the equation a coefficient of pressure drop is used.

The equation with pressure drop is

$\Delta P = K_L . \frac{1}{2} . r. V^2$

By using the experiment results in table 5:1 you can get the coefficient of pressure drop as 32 and this result was imported to fluent as a boundary condition for radiator.

4.2.3. Mathematical Model Of The Inter-cooler

Because of the same reasons stated for radiator a pressure drop coefficient for inter-cooler is calculated as K_L=16 with the experimental results in table 5.3 and imported to fluent as a boundary condition for inter-cooler

Table 4.3 Experimental results for Inter-Cooler

V (m/s)	ΔP (Pascal)
1,76	31
3,78	143
4,43	196
5,43	295

4.2.4. Mathematical Model of The Fan

After passing the inter-cooler and radiator air through cooling room passes fan and exits the cooling room. The main role of the fan is to help heat rejection process of radiator and inter-cooler. By increasing the velocity of the air passing through heat exchangers fan increases the heat energy rejected from coolant and air coming from turbo charger.

Fan is imported to fluent as a pressure jump boundary condition. The amount of pressure jump in the fan changes as the velocity of the air passing through the fan changes. So instead of importing a constant pressure jump, an equation between the pressure jump and velocity of air has to be imported to mode. To find the equation the experimental results in table 5.3 are used.

The fan used in O403 buses has a radius of R_{Fan} =750mm
And the area of fan can be calculated as A_{Fan} =1.767mm^3

After calculating the area of fan we can get the velocity of the air passing through the fan for different amounts of air mass flows through the cooling room.

Table 4.4 Experimental results for fan

Mass Flow Inlet (m^3/s)	Velocity of Air (m/s)	Pressure Jump (Pascal)
4	1664,19	2,26
6	1506	3,39
8	1270,98	4,52
10	959,09	5,65
12	570,34	6,79
14	104,74	7,92

The equation for the curve which is found by using the results on table 5.3 is,

$\Delta P = 1700 + 30v - 30v^2$

This equation is imported to fluent as a boundary condition for fan.

5. FLUENT RESULTS AND OPTIMIZATION

After the developing the geometrical and mathematical model the result is imported to fluent program. For a given mass flow inlet (which is 4 kg/s) the results are calculated for two different installation angels of fan and these models are called Model-1 and Model-2. In Model-1 the fan has the same installation angels with the radiator and intercooler. (All of these parts are installed with a 90 C angle to airflow). But in the Model-2 the radiator and intercooler are installed similar with the first model but the fan has a 3,75 C slope.

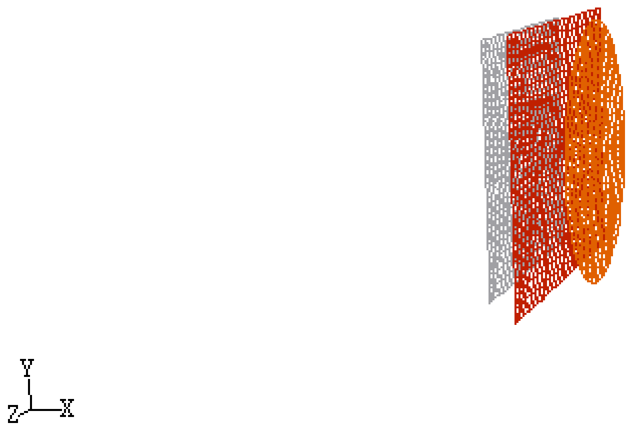

Figure 5.1 The installation of the cooling parts in Model-1

Figure 5.2 The installation of the cooling room parts in Model-2

Because the presence of the batteries and air filter in the cooling room which have resistant effects to airflow, the CFD results for two models are different from each other. In figures 5.3 and 5.4 you can the difference in velocity path-lines for both of the models.

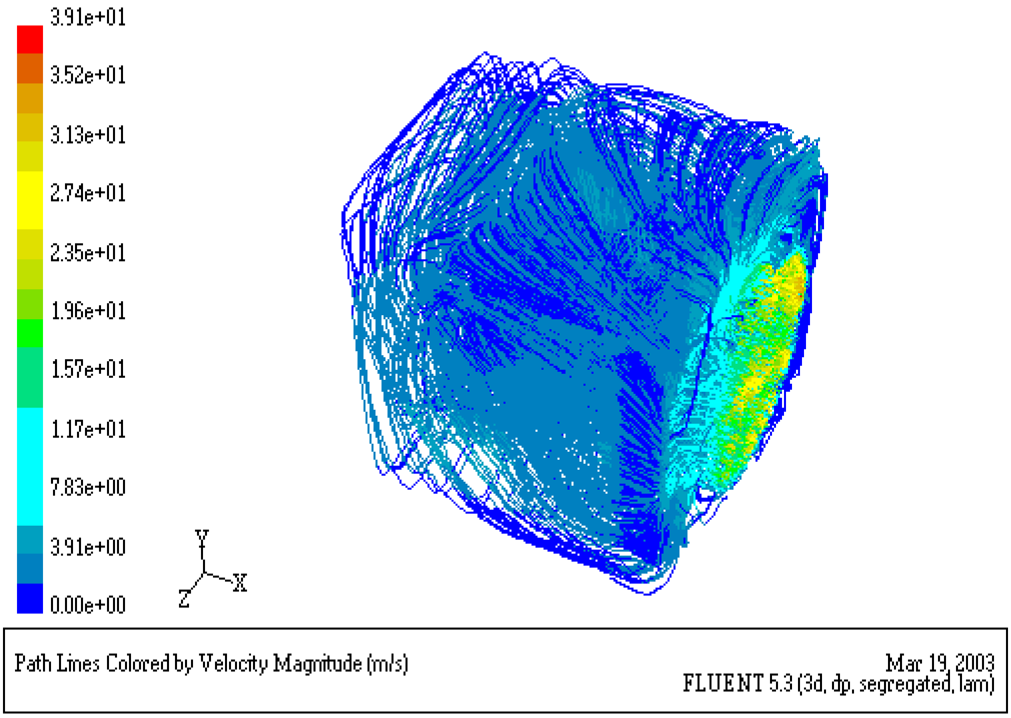

Figure 5.3 Path lines of velocity magnitude for Model 1

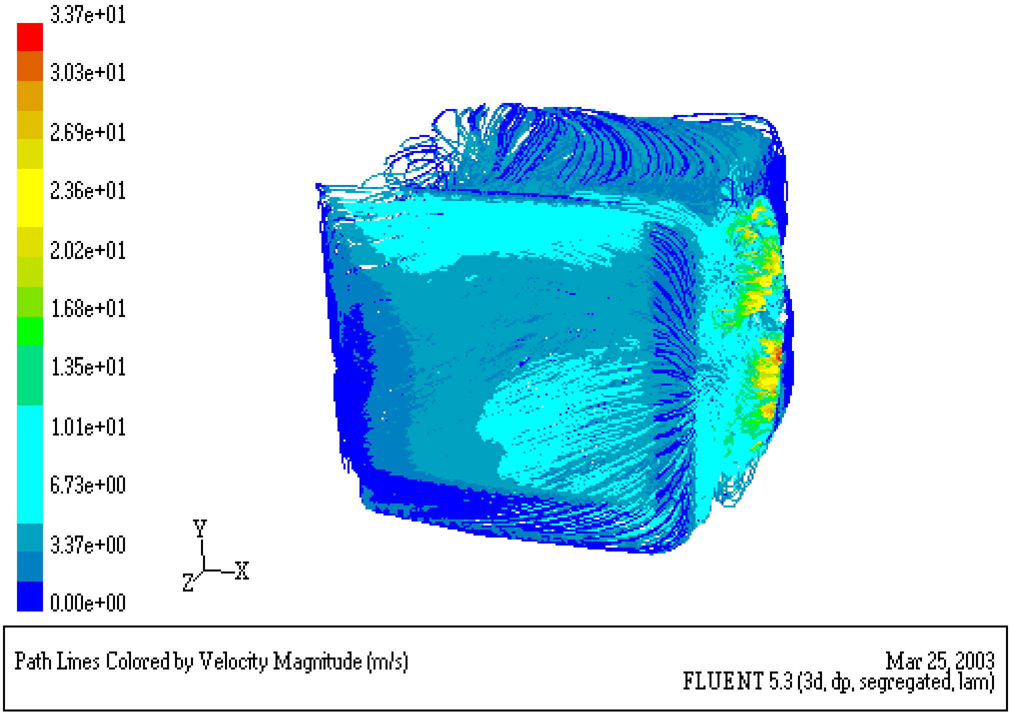

Figure 5.4 Path Lines of velocity magnitude for Model-2

As you can see from the path line of velocity vectors figures there are no big differences of airflow in the cooling room. But in the following velocity versus position diagrams for both of the models we can easily see the differences of airflow in radiator and intercooler.

In the following figures you can see velocity diagrams for different position changes. In some of them (y) position is constant which corresponds to the half of the radiator and intercooler and the change is in (z) direction. And in the other ones (z) position is constant which corresponds to the half of the radiator and intercooler while the change is in (y) direction. We can effectively see the results of the installation angle for the fan from these diagrams.

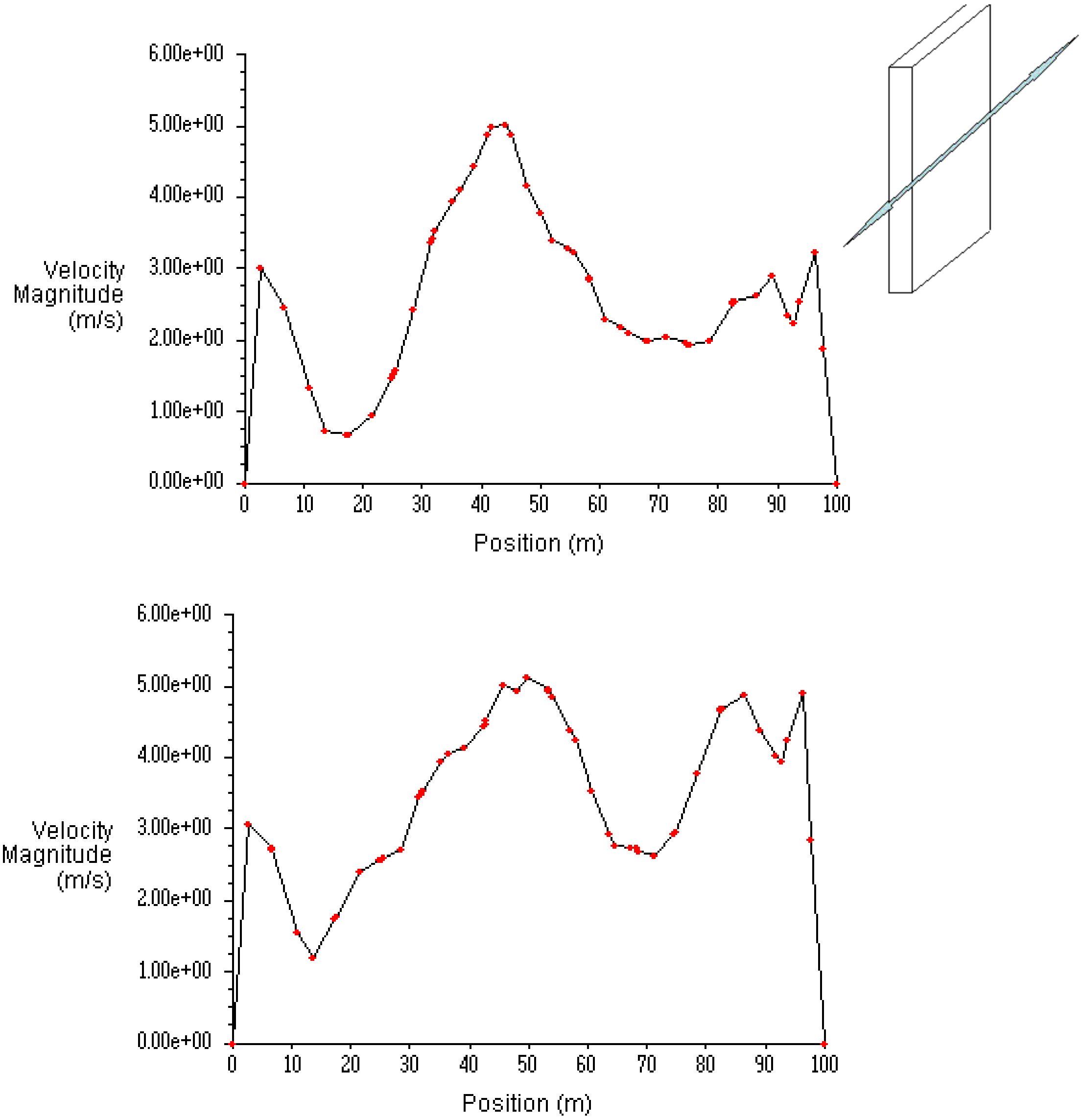

Figure 5.5 Velocity diagram for constant (y) and changing (z) for intercooler in Model-1 and Model-2

From the diagrams the effect of installation angle for fan is clearly seen in airflow through intercooler. The second diagram shows the results for Model-2 in which the fan is installed with a 3,75 C slope. The velocity changes for both conditions seem similar. But this installation slope of the fan decreased the resistant effects of batteries and air filter and as a result the mass flow of air through the intercooler is increased for intercooler. The installation slopes seems more effective in increasing the mass air flow in the part where it is nearer to velocity inlet face. (z =50-100)

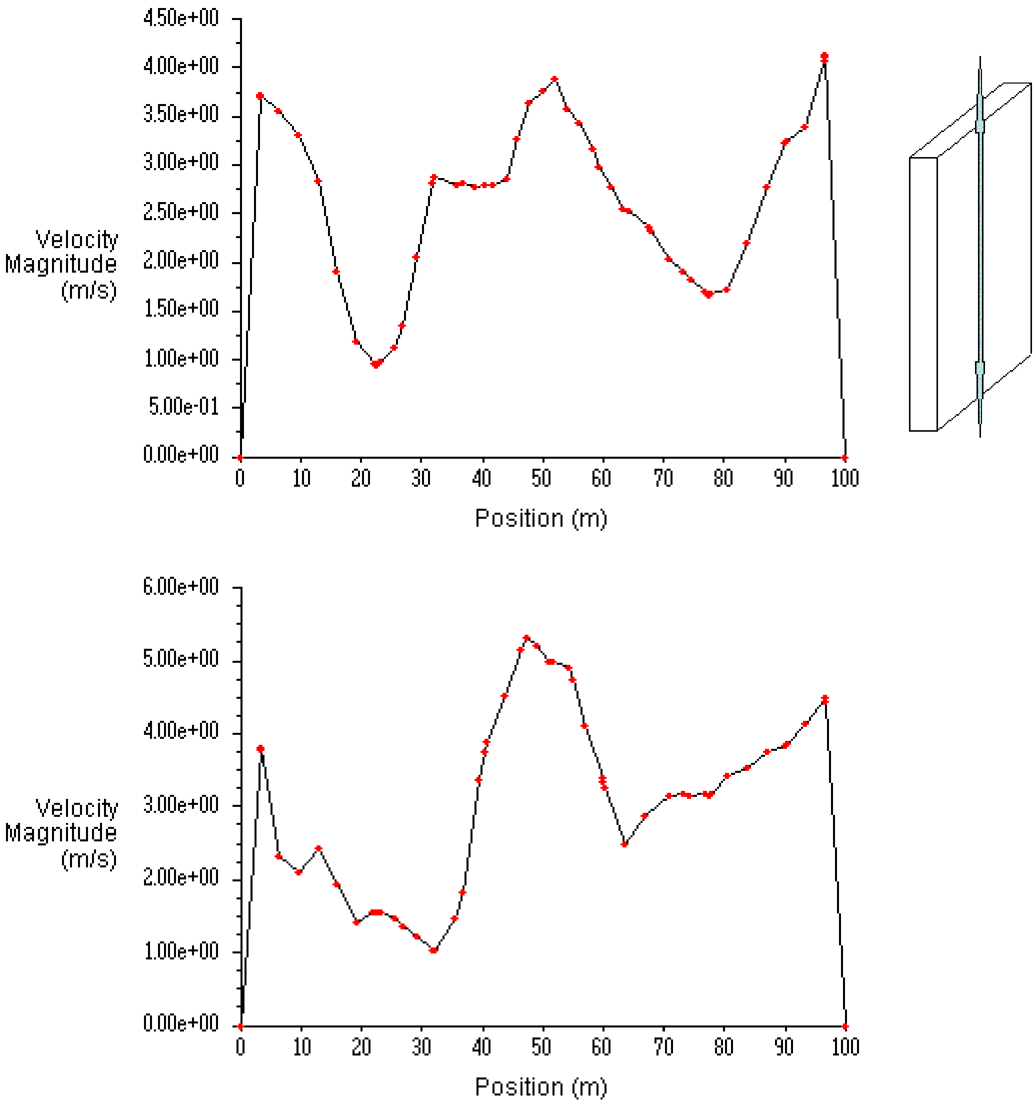

Figure 5.6 Velocity diagram for constant (z) and changing (y) for intercooler in Model-1 and Model-2

From the diagrams it is seen that the airflow in Model-1 has sharper changes than Model-2. These sudden sharp changes of air velocity can have bad effects to intercooler capacity. But the aim of this study is to make an optimization of cooling parts locations not to examine properties of cooling parts. Again the installation of the fan with a slope of 3,75 C has increased the mass flow of the air passing through changing (y) at a constant (z) of intercooler. This results better heat rejection process in intercooler.

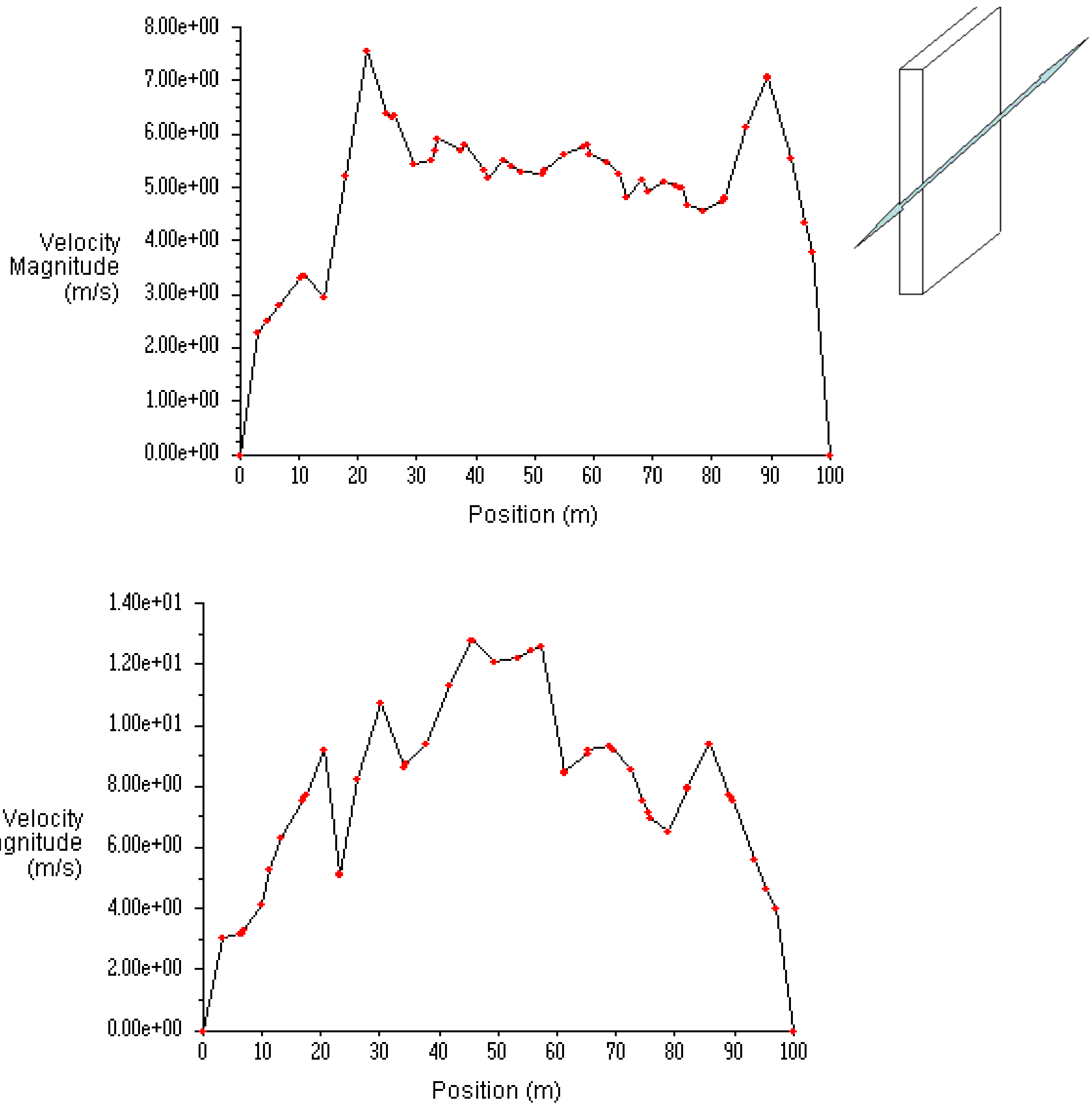

Figure 5.7 Velocity diagram for constant (y) and changing (z) for radiator in Model-1 and Model-2

The continuity of airflow is better in Model 1tha in Model-2 for radiator as can be seen from the diagrams. But when the fan is installed with a slope of 3,75 C the maximum velocities of air passing through the radiator is bigger. The sharp changes of velocity in Model 2 could effect the cooling properties of radiator in a bad way. But as the mass air flow passing through radiator increase without thinking of the radiator properties the heat that is rejected from coolant will also increase. And so again the installation slope has advantages in mass flow of air through the radiator like intercooler.

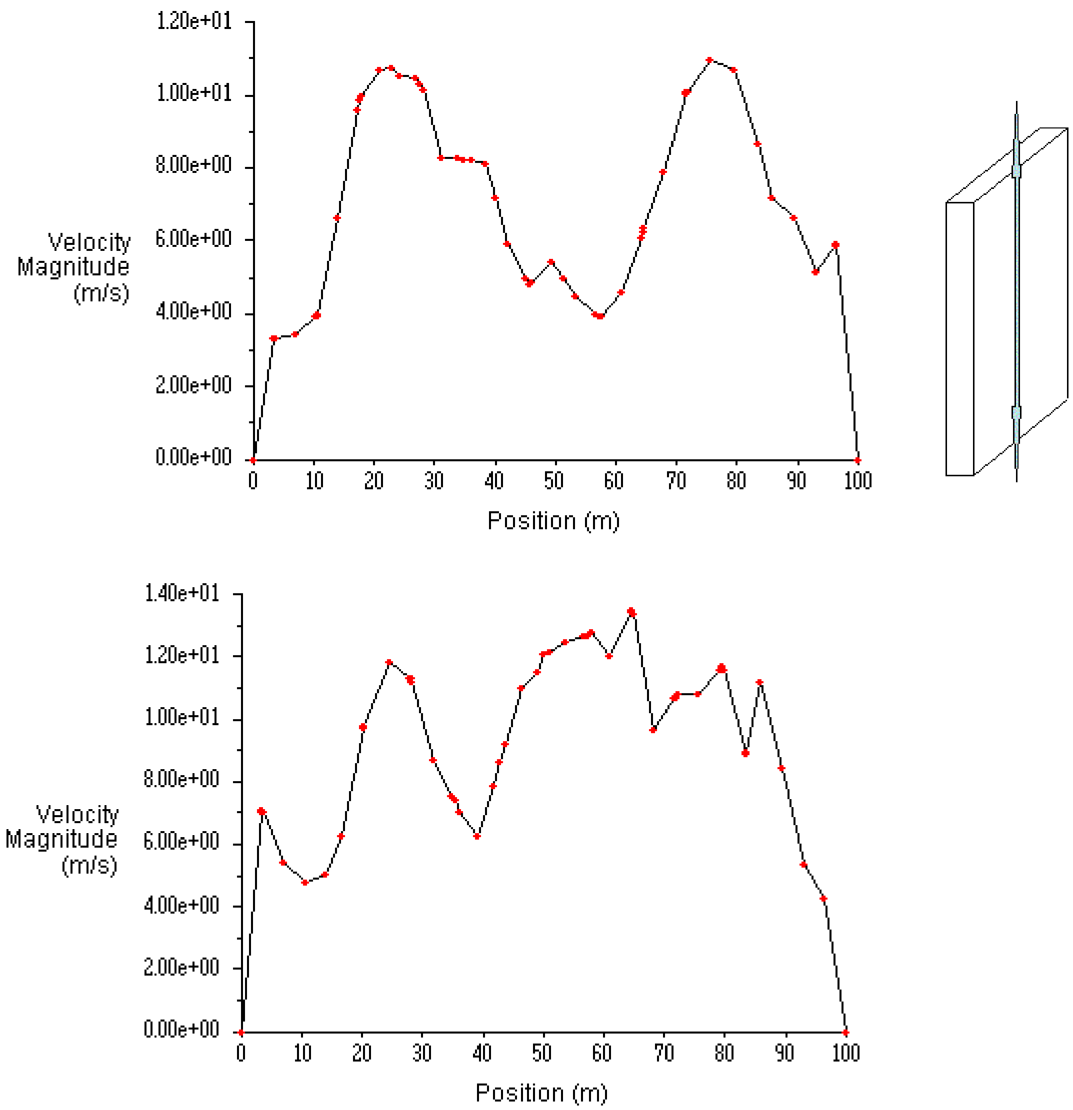

Figure 5.6 Velocity diagram for constant (z) and changing (y) for radiator in Model-1 and Model-2

The situation is same for the velocity changes in radiator for constant (z) and changing (y) for two of the models. Again the continuity properties are better in Model-1 but the maximum air velocities reached in model 2 is higher. So installing the fan with a 3,75^{o} slope is better for all of the situations

From these Fluent results we can state that installation of the fan with a slope of 3,75^{o} is decreased the resistant effects of the batteries and air filter while it increased the mass airflow through the radiator and intercooler. The more air passing through the cooling parts means the more cooling capacity for a constant volume. And as an optimization of the cooling parts installation we can state that installing the fan with a 3,75^{o} slope has increased the cooling capacity of the system.

6. CONCLUSION

The aim of this study was to make an optimization for the installing locations of the cooling parts in cooling room in order to reach higher cooling capacities for same cooling parts. The cooling room of an O403 bus examined with this aim.

First of all the geometric model and the mathematical model for the cooling room are developed. The assumptions that will not have major effects on the results for these models stated. The final model developed than imported to Fluent Program. The results for two cases examined. In the first model the fan was installed perpendicular to airflow and cooling room base. But in the second model the fan was installed with a $3,75^o$ slope.

From the fluent results the air passing through the intercooler and fan in Model-2 is higher. This means for the same volumes of cooling parts the design of Model 2 is better. This situation is a result of the resistant effects of parts some of the parts like air filter and batteries in the cooling room. Installing the fan with a $3,75^o$ slope decreased these resistant effects and increased the mass of air passing through the intercooler and radiator.

This solution was reached by only using a CFD program, without making any prototypes or experiments so this study can be a good example for the advantages of using simulation tools in design of cooling systems

REFERENCES

Papers

[1] **Arıcı,O., Jonson,J., and Kulkarni,A** 1999. The Vehicle Engine Cooling System Simulation Part1- Model Development, Michigan Technological University, *SAE Technical Paper 1999-01-0240*

[2] **Arıcı,O., Jonson,J., Parker, G., and Lehner,C.** 2001. Design and Development of a Model Based Feedback Controlled Cooling System for Heavy duty Diesel Truck Applications Using a Vehicle Engine Cooling System Simulation, Michigan Technological University, *SAE Technical Paper 2001-01-0336*

[3] **Grafenberger, P., Klinner,P., and Nefischer,P.** 2000. Simulation of Engine Cooling with Coupled 1D and 3D Flow Computation., ATZ Automobiltechnisce Zeitschrift, April 2000

[4] **El-Khabiry,S.** 2001. Simulation Reduces Time Needed to Design Commercial Vehicle Air Conditioners. Bergstorm Climate Systems LLC, *Journal Articles by Fluent Users JA125*

[5] **Daimler Chrysler.** 2001. Motor Kumandası Motor Sistemleri 500,900 Serisi Motorlar. Global Eğitim Notları. Mercedes-Benz Türk A.Ş

[6] **Arıcı,O., Jonson,J., and Kulkarni,A** 1999. The Vehicle Engine Cooling System Simulation Part2- Model Validation Using Transient Data, Michigan Technological University, *SAE Technical Paper 1999-01-0241*

[7] **Ap,N.,**1999.A Simple Engine Cooling System Simulation Model. Valeo Engine Cooling, Michigan Technological University, *SAE Technical Paper 1999-01-0237*

Curriculum Vitae

Name: Mehmet Levent Timur
Address: Şair Mehmet Akif Sok. Yüksel Apt. No9/8 Fındıkzade İstanbul
Telephone: Home: 212 6321290 GSM: 533 3252163
Date of Birth: 11 December 1978
Nationality: Turkish
Marital Status: Single

Education:
2000 - 2003

Istanbul Technical Universty Instute of Science and Technology
M.Sc Mechanical Enginnering (Automative Section)
1996 - 2000

Istanbul Technical University Mechanical Engineering Faculty
B.Sc. Mechanical Enginnering
1989 - 1996
Adnan Menderes Anatolian High School

Skills Profile:

Excellent Written and Speaking English
Medium level of Written and Speaking skills of German
Windows NT/98/200/XP Ofiice
Fluent
Autocad

Membership of Professional Bodies:

Asme,TMMOB,Maçka Dans Klubü,

Hobbies and Interests:
Tennis,Basketball,Swimming, Latin Dances