CHALMERS

1D Transient Simulation of Heavy Duty Truck Cooling system – HDEP 16 DST, Euro 6

Master's Thesis in the Automotive Engineering

GANESH RAGHAVAN

Department of Applied Mechanics
Division of Fluid Dynamics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2012
Master's thesis 2012:49

MASTER'S THESIS IN AUTOMOTIVE ENGINEERING

1D Transient Simulation of Heavy Duty Truck Cooling system – HDEP 16 DST, Euro 6

GANESH RAGHAVAN

Department of Applied Mechanics
Division of Fluid Dynamics
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2012

1D Transient Simulation of Heavy Duty Truck Cooling system – HDEP 16 DST, Euro 6
GANESH RAGHAVAN

Master's Thesis 2012:49
ISSN 1652-8557
Department of Applied Mechanics
Division of Fluid Dynamics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: + 46 (0)31-772 1000

Chalmers Reproservice / Department of Applied Mechanics
Göteborg, Sweden 2012

1D Transient Simulation of Heavy Duty Truck Cooling system – HDEP 16 DST, Euro 6

Master's Thesis in the *Automotive Engineering*
GANESH RAGHAVAN
Department of Applied Mechanics
Division of Fluid Dynamics
Chalmers University of Technology

ABSTRACT

In future and also in the present time, with the focus on minimizing environmental impacts, the truck industry faces a big technological challenge in terms of meeting statutory emission legislations and also on satisfying the ever increasing demand of customers in terms of minimizing the fuel consumption. There are other challenges in terms of having a short development time and reducing the overall development cost.

All the above stated challenges requires measures in terms of how computer simulations can be used to better represent a system, how different concepts can be tested, how the overall system can be tested in particular system working environment which ultimately will give a short development time with minimum cost.

This thesis work basically answers the above questions in a holistic manner by considering how the truck cooling system be modeled using different CFD tools like AMESim and GT Cool to understand how different performance parameters of a cooling system vary for a steady and transient driving cycle.

In this thesis work, the cooling system model has been developed for an ongoing project in Volvo Powertrain AB. The model has been developed for 16L DST, 750 Hp, Euro 6 heavy duty truck engine with other auxiliary components like, air compressor, transmission oil cooler, cab heater, urea heater to mention a few. The model has been developed such that it can run on both steady and transient cycles by changing few elements in terms of how the input is given to the model. One of the aims of this thesis work was to evaluate the two tools mentioned above in terms of workability, implementability and reliability.

Results in terms of pressure drop, mass flow rate, heat transfer rate, thermostat valve fluctuation etc. have been compared for above mentioned tools. It is pointed out that since the model has been developed for an ongoing project, the validation of the model by performing actual tests couldn't be performed because of the unavailability of the engine.

In the end certain conclusions have been drawn out in terms of cooling system performance and how effective the tools were in simulating the cooling system.

Key words: Cooling system, 1D simulation, Transient analysis

Contents

Preface

This report is the result of the master thesis work carried out in the Fluid management Group at Volvo Powertrain AB, Göteborg. This thesis fulfills the partial requirement for the Master degree in ‘Automotive Engineering’ at Chalmers University of Technology, Göteborg, Sweden.

The main motivation behind performing this thesis work is the fact that, there is tremendous scope of improving the fuel consumption and reducing the emission formation of an engine from the Thermal management perspective of a vehicle. An optimized thermal management system can lead to great benefits in terms of reduced fuel consumption and reduced impact on the environment. An effort has been made to represent how the vehicle thermal management system will behave in transient conditions.

I would like to thank Mr. Ola Styrenius, Manager, Volvo Powertrain AB for giving me an opportunity and all the necessary resources for carrying out this thesis work in his group. I would also like to thank Mr. Dev Sajjan and Mr. Andreas Ljungberg of Volvo Powertrain AB for being my supervisors during the entire period of this thesis work. Their timely and good guidance has helped me finish the thesis on time. I thank Professor Lars Davidson for being my examiner at Chalmers University. I also appreciate the help extended by Mr. Brad Holcomb, from Gamma Technologies Inc, USA and Mr. Brendan Kane from LMS International, UK for helping me out with various aspects of the modeling tool. Finally, I would like to show my gratitude to all the people in Volvo Powertrain AB who have helped me get all the necessary information and for sharing their experiences for carrying out this thesis work.

Göteborg October 2012

Ganesh Raghavan

Notations

1. CAE - Computer Aided Engineering
2. DST - Dual Stage Turbo charger
3. 1D - 1 Dimensional
4. 3D - 3 Dimensional
5. CFD - Computational Fluid Dynamics
6. EGR - Exhaust Gas Recirculation gas
7. HDEP - Heavy Duty Engine Platform
8. CAD - Computer Aided Design
9. IMTD - Inlet Manifold Temperature Difference
10. CAC - Charge Air Cooler
11. LT EGR - Low Temperature Exhaust Gas Recirculation
12. HT EGR - High Temperature Exhaust Gas Recirculation
13. NOx - Nitrogen Oxides
14. LP CAC - Low Pressure Charge Air Cooler
15. PID - Proportional, Integral, Derivative
16. HT - High Temperature

1 Background

One of the challenges of the modern truck industry with respect to transportation and logistics is the requirement of shorter development times and fuel efficient vehicles. Most often, physical testing route is employed for development. But today's focus on cost cutting and overall competitive nature of the industry necessitates the use of computer simulations with minimum number of physical tests to reduce cost. Simulating the whole system in transient conditions or the actual component working condition has its advantages in terms of flexibility in evaluating different concepts in relatively short time and cost and in a realistic manner.

It is also pointed out that over the past few years a lot of work is going on in the thermal management areas as research has pointed out, for meeting future emissions and reducing fuel consumption, the efficient thermal management strategies needs to be put in place. Recent studies have shown that, still there is umpteen scopes for improvement in this area.

As a rule of thumb, in combustion engines only one third of the internal energy of the fuel is converted into useful work which is available at the crankshaft, one third goes into the exhaust and the remaining one third is taken up by the cooling system to avoid overheating of the engine. This heat is transferred to the surrounding air from the cooling system by means of conduction, convection and radiation.

In predicting accurately the overall temperatures, pressure drop, mass flow rate of the cooling system many 3D CFD simulations are required in order to determine air flows, coolant flows through the cooling system package for different vehicle operating parameters. This kind of analysis consumes a lot of time and is to some extent complex and cost wise also on a higher side. The cost and time gets tremendously increased when transient simulations are performed because of the inherent unsteady nature which requires a lot of computational time. Another approach employed in industry is the 1D simulation which consumes less time and is fairly accurate and fast. In this thesis work the 1D approach has been employed in modelling the truck cooling system.

2 Introduction

Recent studies have shown that, improved thermal management can contribute significantly to emission and fuel consumption reduction. In these lines, a lot of effort is being put in many truck manufacturing companies to improve the efficiency of the thermal management system and to get faster result with minimum simulations cost.

Steady state and transient simulations are being performed to analyse how the system works. A transient simulation gives very good results as it depicts the real working condition of a system. It is important to understand how the system behaves under time varying load, speed, air flow, coolant flow etc. and based on this, improvement in the overall cooling system can be brought about.

Another important aspect is to model the warm-up behaviour of the engine and other accessories. It is of significant importance to understand how long for example, an engine takes for attaining its working temperature. It is in this time interval, the engine operates in its least efficient point, which means because of higher friction and non-efficient combustion, the engine's emission and fuel consumption goes up. It is imperative to minimise this time interval by proper thermal management strategy. An effort has been put in to analyse this phenomenon along with transient conditions mentioned above to understand how the overall system functions.

The scope of this report will be limited only to the coolant circuit and analysis will be limited to HDEP 16, DST Euro 6 engine, which is basically a 16L heavy duty engine with dual stage turbo for meeting increased power requirement. Since the engine was in the conceptual stage when this thesis work was carried out, the cooling circuit initially consisted of a dual loop but subsequently the cooling circuit was modified to a single loop because of different heat loads and flow rate on various components. The dual loop circuit has a high temperature circuit consisting of components like the engine, oil cooler, EGR cooler, retarder etc. and a low temperature circuit consisting of components like Transmission oil cooler, Air compressor and a low temperature EGR cooler on the other hand in a single loop cooling circuit all the components are stacked in the single loop.

3 Objectives

The Objective of this thesis is to first build a 1D model using AMESim software package. The built model should be useful for both steady and transient simulation to evaluate different heat transfer parameters of the whole system.

The second stage will be to apply relevant and suitable forced convection heat transfer correlations and implement the warm-up behaviour of the engine and make it work on a driving cycle and to analyse how the overall system cope up with variations of air flow, coolant flow, speed, and loading. All the major contributing factors for heat transfer need to be determined and implemented in the model.

Another objective of the thesis is to implement and evaluate the same model in a different package, GT Cool to evaluate the results and compare which software package depicts real test values. Volvo Powertrain currently uses AMESim for performing all cooling system simulations and to know how GT Cool would work, this study was done. Another important reason for this study on GT Cool was that, all combustion calculations in Volvo are done in GT Suite and it would be a good way forward to simulate different systems in the same simulation environment to take advantage of the synergy between different systems. All modelling results will be verified with real tests based on availability of the engine and test rigs.

4 Method

The initial modelling approach employed considering the objectives, was to build a steady state model for the HDEP 16 DST engine. In order to do this, the whole architecture of the thermal management system pertaining to 16L DST engine was studied and analysed thoroughly. Based on this, relevant modelling elements in the AMESim libraries were chosen. The model was built with the help of the CAD model of the whole system for getting all the geometrical data and it was made sure that, all the bends, restrictions, orifices were carefully built in the model. All the necessary data files for pressure drop of components were collected and were used as input to the model.

The next step was to implement the heat transfer correlations to determine the heat load across each component like the engine, EGR cooler, radiator. Based on this necessary controls were implemented using signals and controls library of AMESim. Similar controls were implemented for the Charge air cooler and the Fan flow.

The heat load to the coolant is represented as accumulated heat load from the engine which is basically a significant one point source. The heat load to components like EGR cooler, radiator etc. are calculated based on the respective fluid temperatures (coolant and exhaust gas) and the mass flow rate. All these controls have been implemented in the super component facility of the AMESim.

In order to calculate the pressure drop in individual components, the pressure drop data from the supplier of those components, and for some components, pressure drop data from in-house tests and simulations were used.

Next in line was to implement the transient behaviour for each component. Correlations for heat transfer were carefully chosen to take into account various factors affecting the physical properties, turbulence of the coolant flow (discussed in detail in the section 9 of this report) in the engine especially as it account for almost 70% of the coolant heat load. Similarly the warm-up behaviour of the engine was modelled using proper convective elements in AMESim library.

Once, the transient simulation was implemented in AMESim, the same concept was implemented in GTCool. The cooling system model was again built in GTCool, although this time around, certain inputs related to combustion and other detailed geometrical data for the components were required. Apart from this certain controls for example, the fan control was implemented using the controls library of GTCool.

A flow chart showing the methodology is shown below in Figure 1.

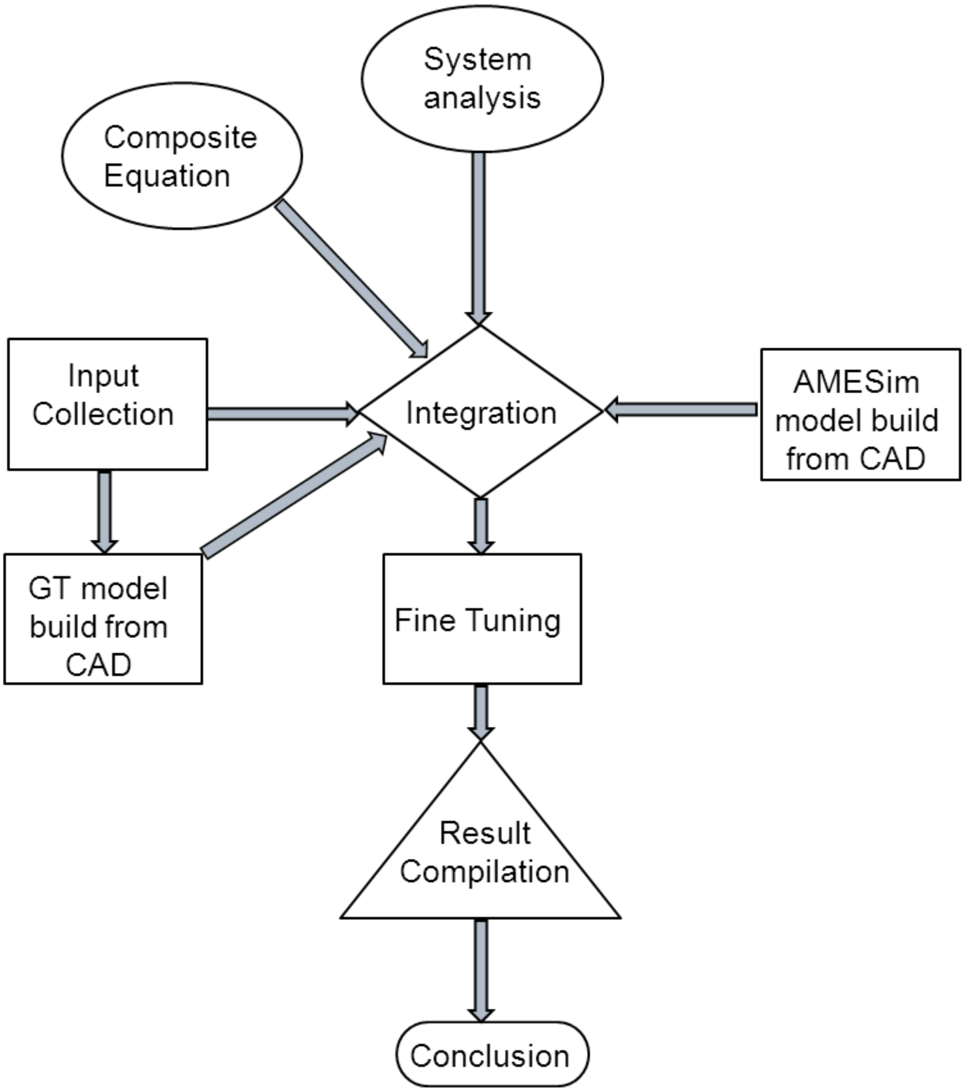


Figure 1: Flow chart showing the methodology employed

5 Limitations and Challenges

5.1 Limitations

The cooling system and engine library in AMESim/GT Cool doesn't cover all the aspects of the cooling, for example, the coolant flowing through complicated geometries in the cylinder block and head and hence certain approximations replicating those complex geometries have to be employed, which does not make the accurate representation of the exact system.

Similarly, the geometries of the cylinder head and block required for calculating the Reynolds's number and in turn for calculating the Nusselt number have been approximated because of the inherent complex nature of the geometries, but much effort has been put to get this approximation as close as possible to actual values to cover the entire coolant flow geometry in the head and block.

Also, the heat load from engine to coolant includes heat transfer both to the lubrication oil and aqueous Ethylene Glycol solution (the coolant). But for this analysis the entire heat load has been assumed to be taken by the Ethylene Glycol solution, as there are no models available in Volvo which determines heat load to both the fluids separately.

Another limitation is the use of pressure drop files for various components. The pressure drop values for different components have been determined at one particular coolant temperature and these files have been used in the model for calculating pressure drop across components for varying coolant temperatures.

The heat load calculation for the charge air cooler (charge air to coolant) has not been implemented as the component is still under development and there are no test performed on the component to get the heat load values. Hence the IMTD value determined does not take in to account the temperature reduction caused by the CAC (charge air to coolant).

The Basic input data, for example, the coolant heat load from engine, charge air cooler mass and temperature, the EGR mass and temperature were available for only few engine speeds and at full load point. In order to create the map, part load values were required to run the model in transient cycle because of which few part load values for each speeds were interpolated. The transient simulation mostly runs on part load conditions and hence the results from transient simulations are not true results and they in a way represent an approximation. Although, the results can be better used to understand how the system copes up for transient cycles and how the engine warm up is affected but it doesn't represent the real values for this particular engine.

Another limitation is the implementation of fan model in GT Cool and AMESim. Because of the lack of availability of time, air side model in AMESim has not been modelled taking into account the physics involved. A very simple strategy has been employed for modelling the air side in AMESim according to a simple map which gives the fan flow based on the speed of fan, ambient temperature, air temperature after the radiator and vehicle speed. But the modelling in GT Cool represents the real physics involved and is more accurate.

.

5.2 Challenges

One of the main challenges faced during modelling of cooling system in AMESim is the calibration of the model. The tool is user friendly, but the calibration process takes quite a lot of time. Also, there were some problems faced during modelling with respect to geometrical data. Once, an element in AMESim is changed from one submodel to another, it changes the geometrical data of the adjacent element back to default and if one is not cautious, the results can be very misleading.

Another challenge was that, since the HDEP 16 DST is an ongoing project, the coolant system architecture has been changed a number of times and hence the model has been changed number of times to accommodate the changes.

GTCool on the other hand, is a large input data driven tool. The models generally require a lot of input data and also, the implementation of certain controls can be a bit complex. During the initial stages of development of system, one may not have all the input data and hence from that perspective it may be difficult to use this tool. Although GTCool allows implementing controls in Simulink and couple with it, but generally coupled simulations are complex in nature.

6 Assumptions

1. Compared to the three mechanisms of heat transfer discussed in section 8, only Conduction and Convection are of significant importance and Radiation plays a negligible part in the whole heat transfer mechanism particularly in the coolant system and hence has been ignored altogether.

2. The combustion models for calculating the gas side heat transfer coefficient are fairly accurate in terms of determining the gas side resistance, but the resistances in certain places for example, the coolant passage in the cylinder head have not been modelled correctly and hence the overall heat load from the engine is not 100% accurate.

3. There is no transient model for the oil circuit and since the oil circuit modelling is not in the scope of this thesis work, the oil side heat load is taken as a constant heat load in transient simulations of the coolant system.

4. The heat load data available as input from the combustion department were only limited to full load points at few speed points and hence the part load values were interpolated to generate the input map.

7 Literature Review

The economic boom in various countries has over the years lead to a phenomenal increase in on-highway heavy duty vehicle operations. With this growth, there is a counter-productive effect with respect to environment. A lot of companies are investing a lot of resources in development of 'Green Technology'. This has come into effect because of stringent emission legislations. Considerable amount of research is happening in the field of alternative fuels, different combustion strategies etc. The field of thermal management has been neglected and has not been kept in pace with development of engine, but lately, there is a significant growth in research in this area. Improvements of the overall thermal management strategy will directly or indirectly improve the fuel consumption and emission [1]. New needs in particular with respect to emissions control like the urea heaters, exhaust gas recirculation (EGR) and strategies involving waste heat recovery once again rely on better thermal management.

The main function of any thermal management strategy of a vehicle is cooling and maintaining temperature across various systems of a vehicle like – engine, lubrication oil, transmission oil, charge air cooler (CAC), EGR cooler, cab temperature etc. This report basically focuses on the thermal management strategies of a heavy duty truck and how this can be improved effectively to reduce the fuel consumption and emission. Over the years how these thermal management strategies have evolved is briefly presented in this section.

System Architecture: The basic engine architecture hasn't undergone any radical change for the past so many years, although certain add-on have been put in place like the cab heater, EGR cooler etc. But even the basic component of a cooling system like the pump, thermostat etc. haven't essentially changed (talking with respect to truck engines) although certain prototypes like the electric pumps and thermostats have been tested but still full-fledged production hasn't been accomplished [1]. There are certain advancements in terms of putting in place a two loop cooling system (for high and low temperature subsystems). An example in this area is the low temperature coolant circuit which can be used for enhancing the CAC efficiency.

Design Optimisation: Normally trucks coming off an assembly line will have different engine specifications with different cooling needs. The engine manufacturer specifies the pump and thermostat specification while the truck manufacturer specifies the radiator and fan specification. In other words the sub components of the same system are custom designed and hence complete optimisation is not necessarily possible [1].

Fan: Fans have been generally engine driven axially which draws up to 10% of the engine power [1]. Hence there have been efforts put on to develop electric fans because of this large power consumption. The electric fan is typically controlled by sensors and is operated in ON/OFF mode which is why it's not typically suitable for the whole working range and condition of the engine and hence cannot control the airflow through the radiator optimally. Better control of the fan working over the overall working range and condition will tremendously improve the air flow according to the required heat dissipation from the engine. Lately, an improvement in this area is the use of clutch type fan, which gives flexibility in disengaging fan where it's not required, apart from this there is an added advantage of reduced Noise and vibration [8]. Apart from electric fans, there have been some developments with fan in terms of implementing a viscous clutch which apart from decreasing power consumption from engine results in reduced noise and vibration [4]. The main aspect of the fan clutch is that the speed varies linearly in proportion to engine cooling load.

Coolant Pump: Typically coolant pumps have been driven mechanically by engine either by gear drive or by belt drive. But lately the belt drive transmission is being used by manufacturers because of higher efficiency. The speed of the pump and thus the coolant flow rate is directly related to engine speed. Hence because of this dependency on the engine speeds, one can say that its functionality is not optimised for the overall thermal management of the engine. For example, this dependency on engine speed will result in a lower coolant flow at low engine speed and high loading condition of the engine, which is usually a region of higher heat load on a comparative basis with low engine speed and low loading point. Because of this dependency there are certain situations, where the combustion for example can occur at temperatures below the optimal engine temperature and can therefore result in higher emission, fuel consumption and poor performance. In recent years there has been an addition to the basic drive mechanism of the pump listed above. In order to drive the pump an electromagnetic clutch has been put in place which reduces the dependency of the pump flow rate to the engine speed which in turn optimises the coolant flow at part load conditions also. Of course, this gets operational with proper control system which selects the best pump speed for any given operational condition [2]. In other concepts a fully electrical pump has also been tried to control the coolant flow rate at each and every operating condition and still a lot of advancements are going on with respect to electric coolant pump and how to effectively integrate this concept into the whole system. All this have significant reduction in pump power requirement which ultimately results in fuel savings.

Fuel Consumption and Emission: A commonly agreed upon goal between office of heavy duty vehicle technologies in the US department of energy and industry is to increase on road fuel economy of class 8 vehicles from 5-7 miles per gallon to 10 miles [1]. To achieve this, several areas need to be focussed apart from engine development, aerodynamics etc. In other words maximum efficiency has to be obtained from the engine. To extract maximum efficiency the engine has to work in optimum temperature. The present thermal management strategies does not allow for accurate control of the engine wall temperatures based on different loading conditions. This has a direct effect on emission generation and fuel consumption.

EGR cooler: EGR have been used for long time as a measure of controlling NOx emissions, but off late as the emission legislations are becoming increasingly stringent, a significant quantity of EGR is used to reduce NOx. Generally when exhaust gas is re-circulated back to the engine, it is cooled. Because of the significant quantities of EGR involved, the increase in heat rejection requirements due to EGR cooling has been estimated to be 50% [1]. Typically EGR is used during the light loading operations, but in future EGR at full load conditions will also be used to meet emissions in addition to certain other strategies [3]. Hence the peak duty of EGR is during light load conditions where the temperatures can reach up to 700 deg C, and the same has to be cooled to around 200 deg C and sent back to the intake manifold. On the other hand cooling systems are generally designed to dissipate heat from the engine which peaks during the full load conditions. Hence coolant flow rate optimised for full load conditions won’t help in reducing the EGR temperatures during light load conditions. Therefore, future thermal strategies should be designed keeping in mind this counteracting phenomenon. In recent years many truck manufacturers have switched to two stage cooling of EGR by incorporating a HT-EGR and a LT-EGR in the high temperature and low temperature circuit (dual loop coolant circuit) respectively to enhance the efficiency of the EGR cooler, so that EGR at correct temperature can be circulated to the engine.

Dual loop coolant system: In recent years several approaches have been taken in terms of basic architecture discussed above to improve the thermal management system of the vehicle. In this direction a lot of manufacturers have successfully implemented the dual loop coolant circuit to improve the thermal management of the vehicle [9]. Although, it has been implemented successfully in terms of architectural synergy with the old system, but there is still a lot to be done in terms of implementation of control strategy. In the dual loop system basically the overall architecture is divided into a low temperature circuit with a separate radiator which takes care of CAC, EGR cooler (in case of LT-EGR and HT-EGR cooler), transmission oil cooler etc. and a high temperature circuit with a separate radiator taking care of the engine heat dissipation, oil cooler, HT-EGR cooler, cab heating etc. Since a lot of subsystems are involved, the control strategy becomes a little complex and a lot of development work is going on in this area.

Thermostat: Over the years thermostats have been basically of wax type. Wax type configuration enables the bypass of the radiator route based on the outlet temperature of the coolant from the engine. Careful studies on thermostats have revealed that this has certain inherent mechanical disadvantages in terms of hysteresis which in turn fluctuates the lift of the valve for some period of time before it gets into steady state conditions. Another aspect in which thermostat plays a vital role is the engine warm-up time. Ideally, the engine should come to its operating working temperature in as little time as possible as it will help in reducing frictional losses, but because of certain physical constraints this time should be minimised as much as possible. There are many prototypes developed in terms of a solenoid operated valve, which negates the above reasoned aspects of the wax type thermostat, but there are certain companies for example Volvo, which has a successful production type solenoid operated thermostats (for their Euro 5 onwards vehicles) which is used in the oil circuit [9].

Radiator: The direct effect of the increase in engine capacity is on the radiator size, as more heat needs to be dissipated as the engine size increases. The governing parameter affecting the design of a radiator is packaging. For a long time now, radiators have been placed in front of the engine vertically because of packaging constraints. Essentially all the radiators are single pass type and with the development of two loop circuits, two radiators of different sizes have been placed in front. There have been many experiments conducted at various levels with respect to inclination and covering the radiator with ducting and each of these have shown considerable improvement in the heat transfer capacity and air flow rates respectively. But incorporating these ideas in a heavy duty truck engine is a complex and a challenging task because of the above mentioned reasons. But still an attempt must be made in this front to see how significant effect does these ideas on the overall efficiency of the engine.

Factors affecting Heat Transfer Coefficient: Conventionally for pipe flows, very simple heat transfer correlations have been used, for example the Dittus-Boelter correlations. Certain studies have been performed particularly in combustion engines where Dittus-Boelter correlations have been used to predict the heat transfer coefficients. It was found out that Dittus-Boelter correlation under predicts the heat transfer coefficient by around 50%. Investigations revealed that factors like the surface roughness, fluid viscosity variations, under developed flows etc. were responsible for the mismatch. A composite correlation taking into account the above mentioned factors were added to the standard Dittus-Boelter correlation resulting in composite correlations which gives a significant improvement in predicting convection heat transfer correlations [5]. These factors have been discussed in detail in section 9 of this report which in a sense forms the basis for carrying out this thesis work.

8 Heat transfer theory

Heat transfer is the thermal energy in transit caused by a temperature difference. Heat transfer can occur basically in three different ways, namely – Conduction, Convection and Radiation. It can occur either alone or in combination depending on the object under consideration. When the object under study is stationary and there is a temperature gradient in it, heat transfer through conduction takes place. Similarly, the heat transfer which will occur between a moving fluid and a stationary object is significantly convection. Although in reality, convection doesn't exist alone and some amount of heat transfer takes place through radiation.

Conduction

''Conduction is the transfer of energy from high-energy particles of a substance to the adjacent low-energy particles as a result of interactions between the particles. In solids, conduction is the result of the vibrations of molecules and the energy transport by free electrons. The amount of energy transferred depends on the internal temperature difference in the volume, cross section area and thermal conductivity of the material. The heat transfer rate can be described by'' [10]:

$$Q_{Cond} = k * A * \frac{dT}{dx}$$

Where Q_{Cond} is the rate of heat transfer, k is the thermal conductivity, A is the cross section area, dT is the temperature difference between the layers and dx is the distance between them.

Convection

''Convection is the energy transfer between a solid surface and an adjacent fluid that is in motion; it is the combined effect of conduction and fluid motion. Convection can be natural or forced. Natural convection is a form of conduction between the volume and the stationary fluid and occurs because of the density differences due to temperature change in the fluid. Natural convection can be described by'' [10]:

$$Q_{conv} = h * A_s * (T - T_\infty)$$

Where Q_{conv} is the heat transfer rate, h is the convection heat transfer coefficient, A_s is the surface area, T is the volume temperature and T_∞ is the fluid temperature.

Radiation

‘’All bodies with a temperature above absolute zero emit thermal radiation. The energy emitted is in the form of electromagnetic waves and transfers heat from the volume to the surrounding environment. Radiation depends on the surface area of the volume, the temperature difference between the volume and the surrounding environment and the emissivity of the material. The emissivity is a measure of how close the material is to an ideal surface in terms of a maximum radiation rate. The net heat transferred by radiation can be described by’’ [10]:

$$Q_{rad} = \varepsilon * \sigma * A_s * (T^4 - T_\infty{}^4)$$

Where Q_{rad} is the net heat transfer rate, ε is the emissivity, σ is the Stefan-Boltzmann constant, A_s is the surface area, T is the volume temperature and T_∞ is the surrounding air temperature.

9 Factors Affecting heat transfer

As a general case, for calculation of heat transfer coefficient through a duct either rectangular or circular, the 'Dittus-Boelter equations' have been used which can be applied only when certain criteria's are fulfilled, for example the flow has to be fully developed, the difference between the bulk temperature and the film temperature has to be less than 5.6 deg C etc. But the same when applied to water jacket of the internal combustion engine showed a difference of around 20% from actual heat transfer coefficient calculated experimentally [5]. The water jacket in the internal combustion engine poses certain conditions where the 'Dittus-Boelter equation' does not satisfy and hence the original equation has to be modified to be able to apply in the water jacket. The conditions are listed below which must be taken for suitably modifying the original equation.

1. **The Fluid Dynamic Entry Length:** When fluid enters a properly shaped duct, its velocity changes initially and after some distance down the duct, the velocity profile becomes steady or in other words, the flow is called fully developed. The length over which, this velocity is unstable is called as the fluid dynamic entry length. The 'Dittus-Boelter equation' is applicable to fully developed flow. The molecules of the fluid in contact with the wall have zero velocity (no-slip condition) which slows down the molecules slightly far away from the wall. The velocity of the molecules far away from the wall remains unchanged. The result is the development of the fluid dynamic boundary layer where the velocity increases with the increasing distance from the wall. In the entrance of the water jacket, the boundary layer is thinner than in the developed region giving an enhanced heat transfer [5].

$$\frac{Nu}{Nu_{fd}} = 1 + 23.99\ Re^{-0.23} \left(\frac{x}{D_h}\right)^n$$

Where,

$$n = 2.08 * 10^{-6} Re - 0.815$$

In other words, the above equation can be written as below:

$$Nu = (Dittus\ Boelter\ Equation) * \left[1 + 23.99\ Re^{-0.23} * \left(\frac{x}{D_h}\right)^n\right]$$

Where x is the upstream entrance length.

2. **Unheated Starting Length:** In addition to the velocity profile, there exists a temperature profile where the temperature of the fluid changes with distance along the

duct. This temperature reaches a steady value at some distance down the duct. It's important to consider the entrance effects for both the thermal and fluid dynamics as both starts to develop at different locations. The entrance effect because of the thermal boundary layer development downstream of the duct leads to an enhanced heat transfer coefficient. The length over which the thermal boundary layer has not developed is called the unheated starting length. When it comes to the water jacket in an internal combustion engine, the coolant flowing through the duct starts experiencing increase in its temperature right from the start of the coolant duct cover which is basically the starting point of the first cylinder of the engine and hence, this particular factor will not apply to the internal combustion engine [5].

$$Nu_x = \frac{Nu_{x=0}}{\left[1 - \left(\frac{x_0}{x}\right)^{\frac{9}{10}}\right]^{\frac{1}{9}}}$$

3. **Surface Roughness:** The 'Dittus-Boelter equation' applies specifically to ducts which are smooth. The roughness of the surface has very significant effect in terms of the flow patterns wherein it helps in increase of turbulence. The increase in heat transfer is enhanced by two modes – firstly, it increases the surface area and secondly it increases the turbulence and hence the convective heat transfer coefficient increases [5].

$$Nu = (0.023 * Re^{0.8} * Pr^{0.4}) * F\left(\frac{\varepsilon}{D_h}\right)$$

Where,

$$F\left(\frac{\varepsilon}{D_h}\right) = 0.091 * \left(\frac{\varepsilon}{D_h}\right)^{-0.125} * Re^{0.363\left(\varepsilon/D_h\right)^{0.1}}$$

In other words, the above equation can be written as below:

$$Nu = (Dittus\ Boelter\ Equation) * \left[0.091 * \left(\frac{\varepsilon}{D_h}\right)^{-0.125} * Re^{0.363\left(\varepsilon/D_h\right)^{0.1}}\right]$$

4. **Fluid Property variation with temperature:** Strong heating in ducts results in an increase of the fluid temperature which is close to the wall. The fluid properties are very susceptible to temperature change. For example, an increase of temperature from 100 to 150 deg Celsius results in a change of almost 70% in the viscosity value for 50-50 antifreeze water. The reduction of viscosity with increase in temperature causes increase in turbulence, reduction of the boundary layer thickness which in turn results in a higher heat transfer. Because of this temperature sensitivity, all the fluid properties are evaluated at bulk fluid temperature except the viscosity which is determined at the bulk fluid as well as wall temperatures [5].

$$Nu = (Dittus\ Boelter\ Equation) * \left(\frac{\mu_{bulk}}{\mu_{wall}}\right)^{0.14}$$

Taking into account all the factors mentioned above, the original 'Dittus- Boelter equation' can be written as:

Nusselt Number = (Dittus-Boelter correlation)

* Entrance factor

* Unheated starting length factor

* Roughness factor

* Viscosity loading factor

$$Nu = (0.023 * Re^{0.8} * Pr^{0.4})$$

$$* \left(\left[1 + 23.99\, Re^{-0.23} * \left(\frac{x}{D_h} \right)^{n} \right] \right)$$

$$* \left\{ \frac{1}{\left[1 - \left(\frac{x_0}{x} \right)^{\frac{9}{10}} \right]^{\frac{1}{9}}} \right\}$$

$$* \left[0.091 * \left(\frac{\varepsilon}{D_h} \right)^{-0.125} * Re^{0.363 \left(\varepsilon / D_h \right)^{0.1}} \right]$$

$$* \left(\frac{\mu_{bulk}}{\mu_{wall}} \right)^{0.14}$$

Apart from the composite convection model mentioned above, Sleicher and Rouse proposed a different correlation mentioned below:

$$Nu = 5 + 0.015 * Re^{m} * Pr^{n}$$

Where,

$$m = 0.88 - \left(\frac{0.24}{4 + Pr} \right)$$

$$n = \frac{1}{3} + 0.5 * e^{-0.6 * Pr}$$

However, this correlation has been validated only for Re>50000 and it has certain effects from Prandtl number variations [5].

10 AMESim and GT Cool Tool

10.1 AMESim

AMESim is 1D simulation software for analysing multi-domain system for example, vehicle thermal management, powertrain, internal combustion engines etc. It has a number of libraries for building up models in different domains mentioned above. In this thesis, the vehicle thermal management analysis has been performed and hence all the libraries associated to thermal management have been used to carry out the analysis. AMESim is a virtual tool to design, analyse, and optimize the fluid flow characteristic of a system. As a trait of any virtual tool, AMESim reduces the lead time from synthesis of a product to the time it enters the market by reducing cost and time involved in physical tests [6].

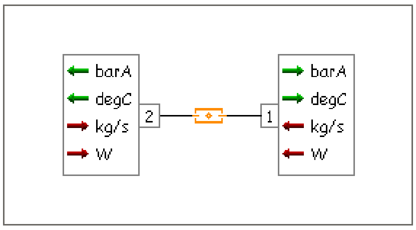
Theory:

All the physical elements in AMESim have been designed specifically for modelling the effects of heat transfer in both steady state and transient conditions. All the elements in AMESim represent a mathematical model and these elements can be connected to form a circuit, which in turn represents an actual system, for example, the cooling system circuit of a vehicle.

Libraries:

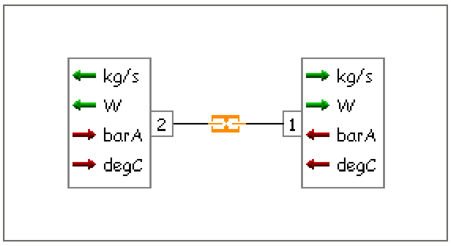
The model in AMESim was built using a combination of 5 libraries namely – Thermal, Thermal hydraulic, Thermal Hydraulic resistance, Cooling system, Signal and Control. The elements of each library have its salient features and the most important ones are described below.

Thermal Hydraulic pipe:



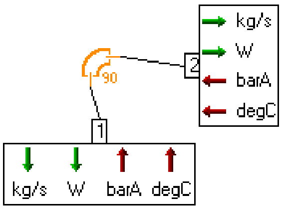
This particular element is used extensively in the model representing the coolant pipes and hoses. This represents the pressure loss in a pipe due to friction. The basic inputs to this element are the diameter, initial temperature, initial pressure, relative roughness, wall thickness etc. The output of this element is the Flow rate, temperature at both inlet and outlet, pressure at inlet and outlet, Reynolds number etc.

Thermal Hydraulic resistance:



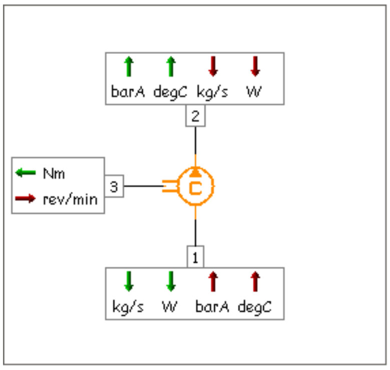
This element is used in the model, wherever there is a sudden restriction in the pipe, to model orifice etc. The main function of this element is that it computes the pressure drop across the inlet and outlet. In the model, this element has been used to represent pressure drop in components such as EGR cooler, Engine block, radiator etc. The input to this element is initial mass flow rate, a file containing a series of data with pressure drop for different mass flow rate. The file is taken from laboratory tests conducted on the component for different mass flow rate and measuring the pressure drop from inlet to outlet. In a simulation, when a particular mass flow rate occurs, AMESim interpolate or extrapolate and calculates the pressure drop.

Thermal Hydraulic Bend:



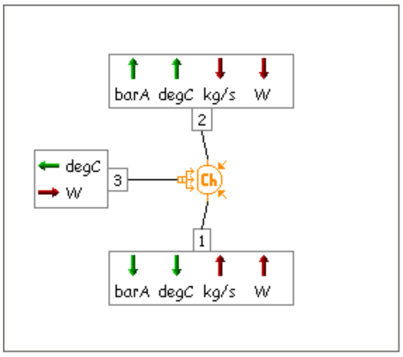
This element is used to simulate bends in the pipes. AMESim library has basically 3 different types of bends – 90 deg, 45 deg and 30 deg apart from a generic element where user defined angles can be used. The main input to this element is the radius of curvature and diameter. As an output, one gets mass flow rate at inlet and outlet, pressure at inlet and outlet, temperatures at inlet and outlet etc.

Centrifugal Pump:



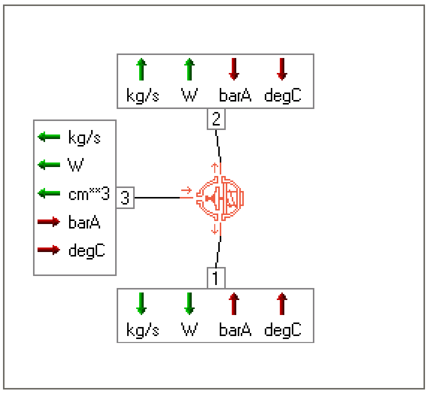
This element represents the centrifugal pumps generally used in automobile applications. The inputs to this element include a file which basically is a series of 3D data representing the mass flow rate, pressure rise and speed. The output is the pressure at the inlet and outlet, flow rate, temperature at inlet and outlet.

Thermal Hydraulic Capacity:



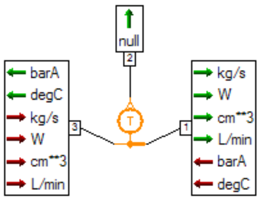
This element represents the volume, for example the coolant volume in the engine block, coolant volume in the radiator etc. The input include, pressure, temperature and the coolant volume. The output includes mass flow rate, Heat load, pressure etc.

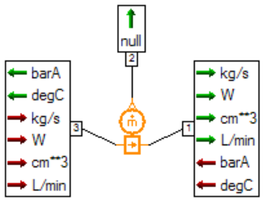
Thermostat:



This element is the exclusive AMESim wax type thermostat. The main input include files which represents the increasing and decreasing slope of the thermostat valve lift with respect to temperature. Apart from this inputs include temperature, cross sectional area at maximum lift at both the bypass port and the main port. It also includes the law which will determine at what rate both the port will open/close. It is also important to specify the time constant for thermostat which basically determines how responsive the thermostat is. The outlet is basically the mass flow rate at both the bypass port and the main port, pressure at each port etc.

Sensors:



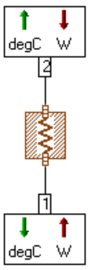


The element above represents the temperature and mass flow sensors. It has the pressure and temperature as inputs. The out puts include mass flow rate, temperature etc.

Signals:

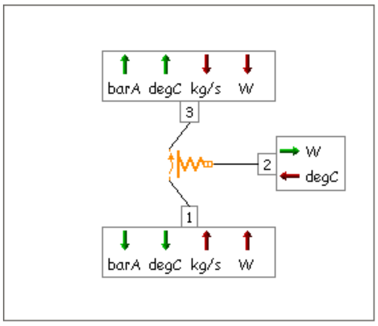
The one to the left is the receiver which receives the signal from a transmitter shown to the right.

Radial Conductive exchange:



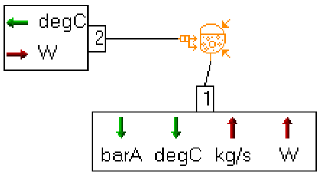
This element is used to model the conductive heat exchange through walls of the water jacket in an engine, through the EGR cooler etc. This element is used in the model particularly to analyse how for example, the engine warm-up takes place in relation to time. The thermal conductivity is computed based on the mean temperature which is the average of the temperature at port 1 and 2.

Internal Flow Convection:



This element is used in analysis of the forced convective heat transfer characteristics of a cooling system. The port 1 and 3 represents the ports through which coolant flows and the port 2 represents the wall which again gets heated by the conduction process for example conduction in the engine wall. The convective heat exchange at port 2 is computed from the Nusselt number by applying appropriate correlation. The main inputs include the hydraulic diameter, the relative roughness, and Reynolds number value for the laminar and turbulent flows.

Thermal Hydraulic Accumulator:

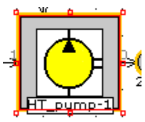


This element is the thermal hydraulic accumulator which is used as an expansion tank in the model. As the expansion tank has two volumes involved (gas and coolant), here also the inputs include the volume of coolant present in the whole accumulator volume.

10.2 GT Cool

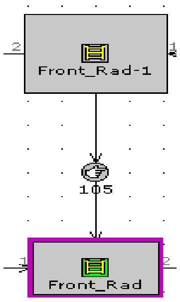
GT Cool is also a 1-D simulation tool mainly used to model the automobile cooling and system. This tool can also be used for both steady and transient simulations. The tool can be used for detailed modelling of the engine and vehicle thermal management [7].

Pump



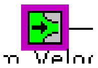
The element is used to model a pump when a map of pump speed, mass flow rate, pressure rise and efficiency is available.

Radiator



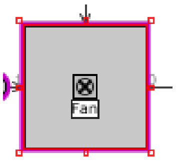
The above element represents the radiator which is basically used to model the heat transfer between coolant on one side and air flow on the other. The top element represents the Master radiator which is used to model the coolant side and Bottom element represents the Slave Radiator which is used to model the air side. The elements can be used to model all the types of flow conFigureations – cross flow, counter flow and parallel flow. The heat transfer is calculated based on the Dittus-Boelter correlation. The HT Radiator, Front Radiator, CAC, EGR Cooler have been modelled using the above element.

End Environment Ram



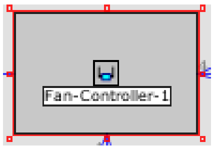
The Above element is used to model the Ram air flow through the vehicle.

Fan



The above element is used to model the fan. The data is given in the form of a map which basically includes parameters like fan speed, mass flow rate, pressure rise and efficiency of the fan. In the model, the fan speed has been controlled using a PID controller.

PID Controller



The PID controller is used in the model to control the fan speed. The PID controller based on the Proportional, Integral gains tries to bring the fan speed to a steady state value. In general the Derivative gain is not used and set to ‘ign’ (ignore) in the element. Basically it tries to bring the input signal to a value such that a target parameter can achieved as soon as possible.

11 Thermal Management System

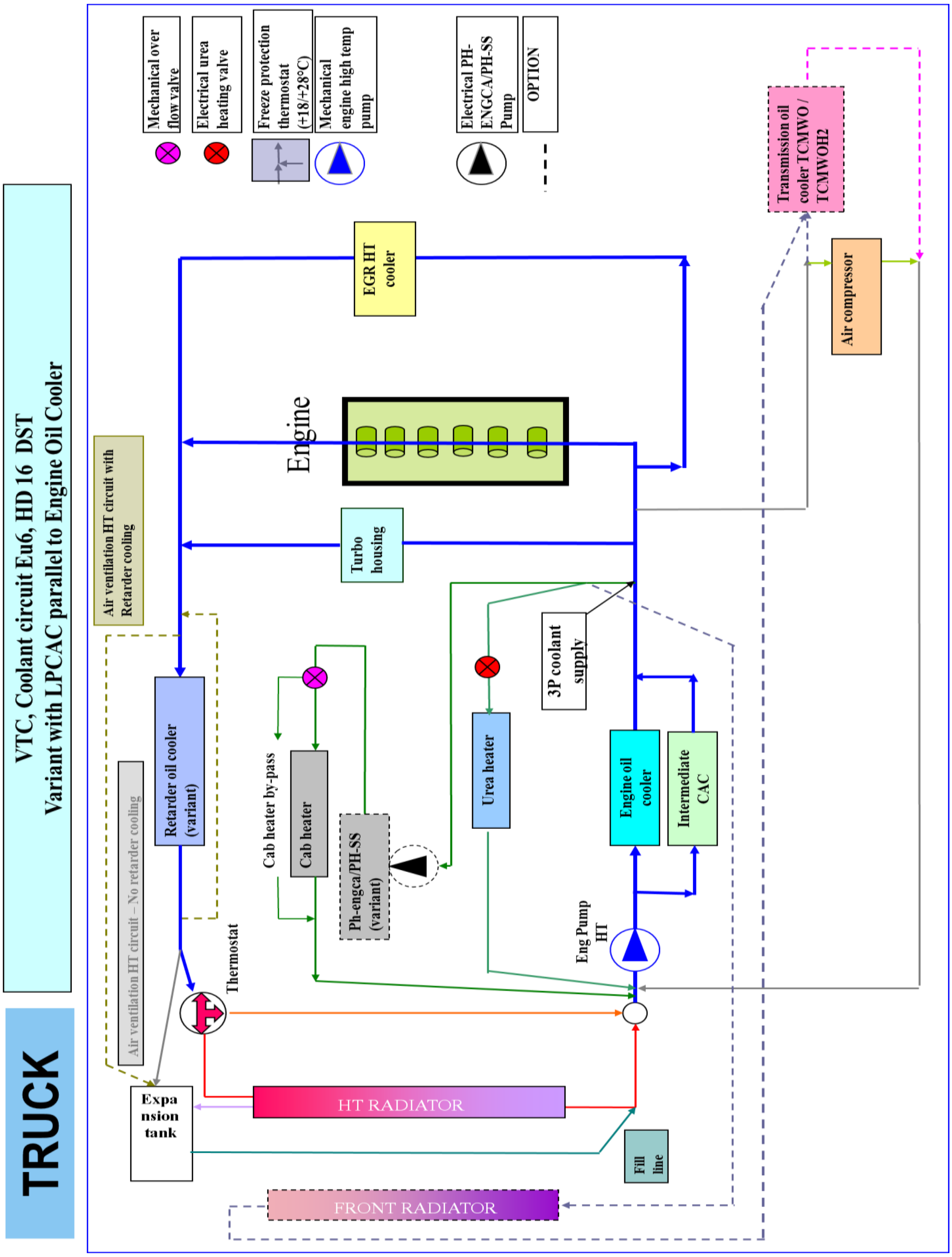


Figure 2: Thermal management architecture of HDE16 EU6

The thermal management strategy takes into account a lot of components like, the engine, EGR cooler (LT and HT), Turbocharger, Cab heater, Urea heater, Retarder, Transmission oil cooler, Air compressor, Engine oil cooler, HT CAC. Complete system architecture of the HDEP 16L DST application for Euro 6 emission norms is shown in the Figure 2.

The version shown above is an application where the transmission oil cooler and air compressor are on the HT circuit. But there is another circuit where, the transmission oil cooler and air compressor are on the LT circuit. The model built in AMESim and GTCool are according to the architecture shown in Figure 2 and the same is shown in Appendix 19.1. Some of the components of the engine is described below along with their pressure drop data.

Engine:

The engine used is a 16 liter engine for Euro 6 emission norms. The development of this engine was initiated because of the higher torque requirements. This higher torque target required the development of a DST. Because of the layout of the engine, a LPCAC was also introduced in the HT circuit, to have a better inter stage cooling. Intermediate charge air cooling will result in a higher torque output, because of increased air inlet density. The pressure drop curve for the engine is shown in Figure 3.

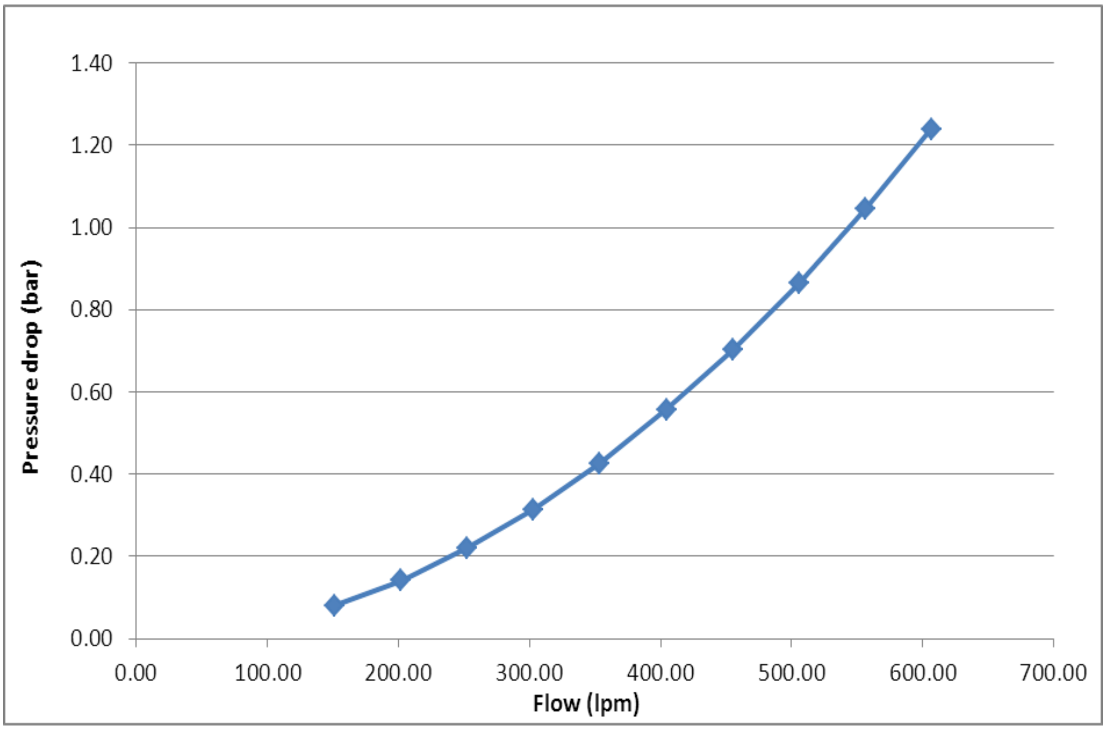
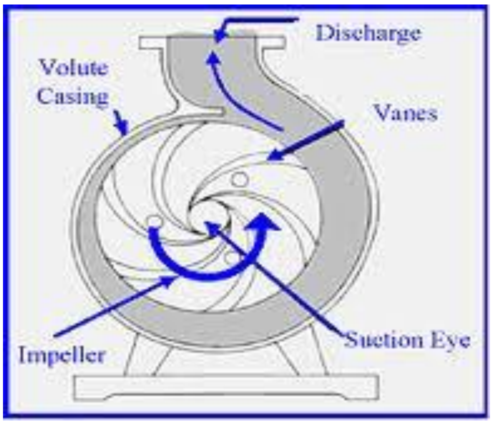


Figure 3: Pressure drop curve over the engine

HT Coolant Pump:

The coolant pump used here is a centrifugal pump which is connected to the engine through a pulley and a belt drive at particular ratio. It is important for the pump to build the necessary pressure to overcome the pressure loss across the circuit and various component of a vehicle. The centrifugal pump converts the kinetic energy into pressure energy.



The pump has two components namely the impeller and the diffuser or volute. The impeller generates the kinetic energy and the diffuser converts into a pressure energy, which means the pump makes the coolant flow at particular flow rate and pressure through the circuit. The pump characteristic curve is shown in Figure 4 where each curve represents different pump speed.

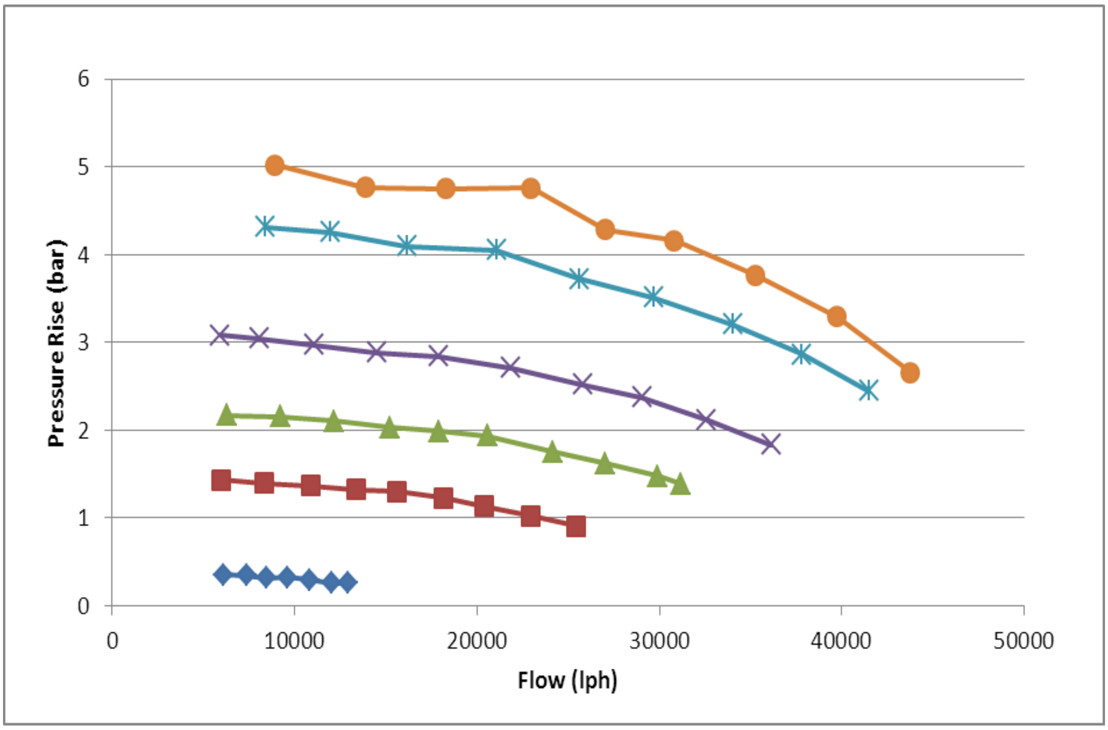


Figure 4: Pump Characteristic curve

Thermostat:

The thermostat used is a wax type, which gets activated by the temperature of the coolant before it enters the inlet of the radiator. The thermostat has 3 ports namely – the inlet coming from the engine cylinder head, the second port is the bypass, which goes directly to the coolant pump and the third port opens up into the inlet of the radiator. The wax in the thermostat controls the opening and closing of the third port, in turn controlling the coolant flow going through the radiator. By controlling the opening/closing of the ports, it prevents coolant to loose excess heat and thereby maintains optimum operating temperature. One disadvantage of wax type thermostat is that it has hysteresis meaning, the rate of valve opening and closing is not the same and hence the port opening and closing won't be at the same temperature. A hysteresis curve is shown in Figure 5.

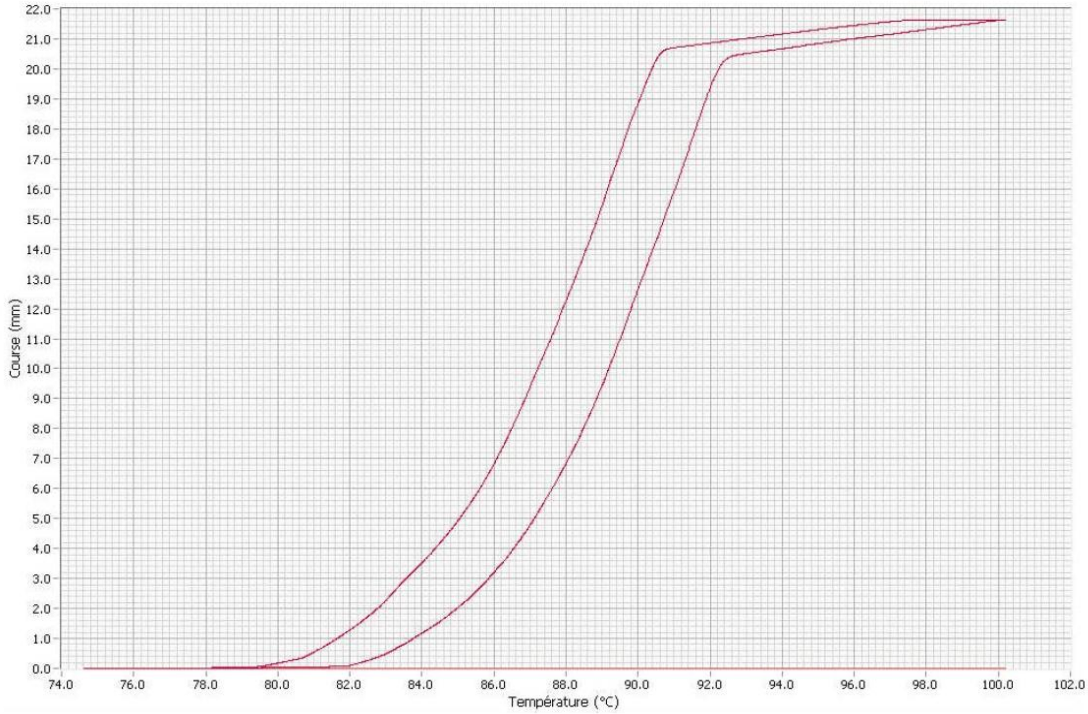


Figure 5: Thermostat hysteresis curve

HT EGR Cooler

The EGR cooler used in the HT circuit, is a dense core cooler. The hot exhaust gases from the exhaust flows through the densely packed tubes of the cooler and the coolant flows outside of these tubes. The whole structure is properly enclosed without any leakages. The flow rate of the hot exhaust gases through the cooler depends on the combustion strategy employed and in turn, determines the heat load on the cooler. The pressure drop curve of the cooler is shown in Figure 6.

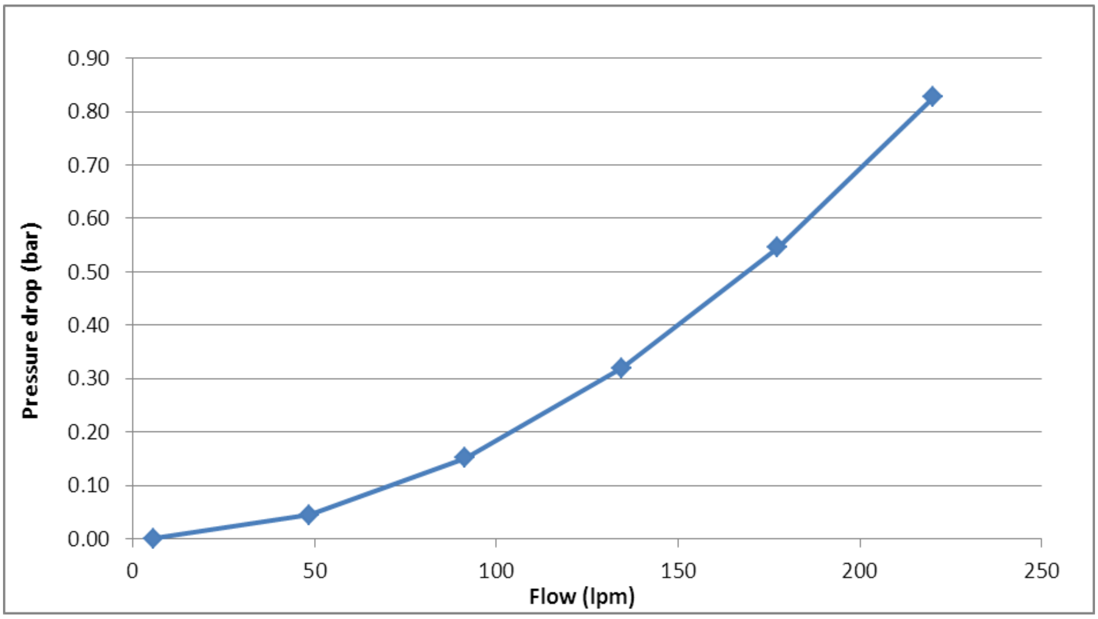


Figure 6: Pressure drop over the EGR cooler

Oil Cooler:

The engine oil cooler used here is a densely stacked tube through which oil flow inside, and the coolant is allowed to flow outside of these tubes. The pressure drop data is shown in Figure 7.

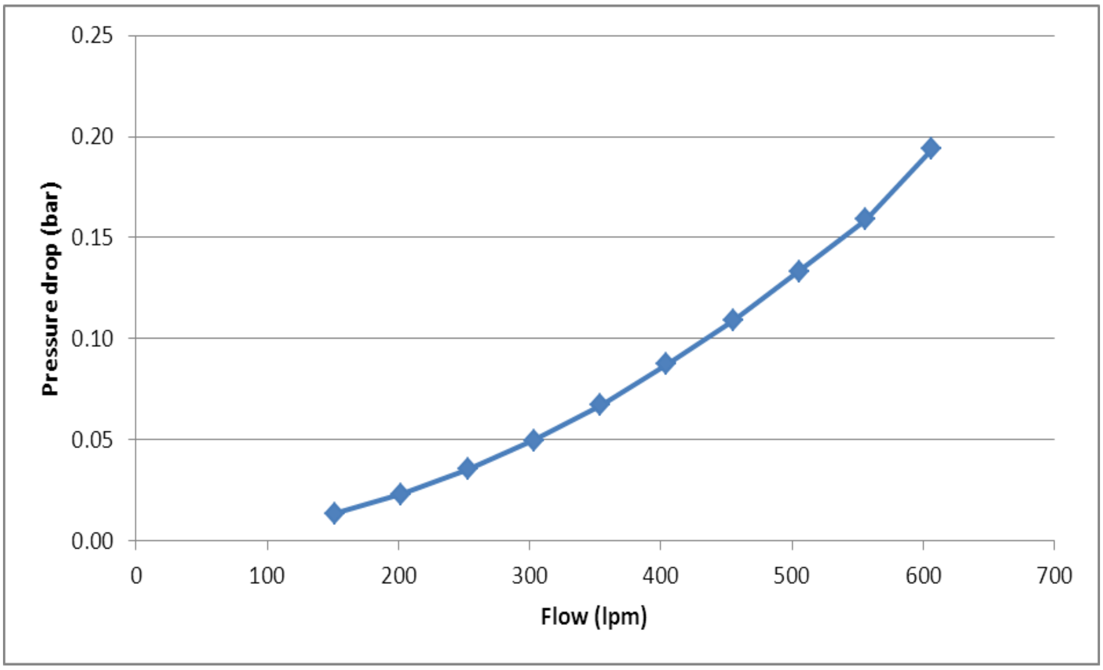


Figure 7: Pressure drop over Engine Oil cooler

12 Input data

Table 1 shows the input data for individual components which are used for the simulation. The engine heat load, EGR mass flow, EGR Temperature, charge air mass flow, charge air temperature. The data which are in bold letters represent the actual input data received from combustion calculation and other part load data were generated using interpolation for transient simulations.

Table 1: Simulation Input Data showing full load and part load data for each speed

Speed	Torque (T)	Heat Load (HL)	EGR Mass Flow (EMF)	EGR Temp (ET)	CAC mass flow (CMF)	CAC Temp (CT)
Rpm	**N-m**	**kW**	**g/s**	**Deg C**	**g/s**	**Deg C**
1800	**T1**	**HL1**	**EMF1**	**ET1**	**CMF1**	**CT1**
	0.75*T1	0.75*HL1	0.75*EMF1	0.75*ET1	0.75*CMF1	0.75*CT1
	0.50*T1	0.50*HL1	0.50*EMF1	0.50*ET1	0.50*CMF1	0.50*CT1
	0.25*T1	0.25*HL1	0.25*EMF1	0.25*ET1	0.25*CMF1	0.25*CT1
1400	**T2= 1.22*T1**	**HL2= 0.80*HL1**	**EMF2= 0.745*EMF1**	**ET2= 1.01*ET1**	**CMF2= 0.89*CMF1**	**CT2= 0.96*CT1**
	0.75*T2	0.75*HL2	0.75*EMF2	0.75*ET2	0.75*CMF2	0.75*CT2
	0.50*T2	0.50*HL2	0.50*EMF2	0.50*ET2	0.50*CMF2	0.50*CT2
	0.25*T2	0.25*HL2	0.25*EMF2	0.25*ET2	0.25*CMF2	0.25*CT2
1200	**T3= 1.22*T1**	**HL2= 0.69*HL1**	**EMF3= 0.715*EMF1**	**ET3= 0.95*ET1**	**CMF3= 0.754*CMF1**	**CT3= 0.84*CT1**
	0.75*T3	0.75*HL3	0.75*EMF3	0.75*ET3	0.75*CMF3	0.75*CT3
	0.50*T3	0.50*HL3	0.50*EMF3	0.50*ET3	0.50*CMF3	0.50*CT3
	0.25*T3	0.25*HL3	0.25*EMF3	0.25*ET3	0.25*CMF3	0.25*CT3

As pointed out earlier, the heat load on components like HT EGR, Radiator, have been calculated using the supercomponents in AMESim wherein controls using Simulink has been implemented which calculates the heat load based on the mass flow rate and temperature of the respective fluids. On the other hand in GT Cool, specific elements representing radiators have been used which calculates the heat transfer in itself and no specific controls have been used to calculate the same.

Apart from the data shown in the tabular column above, other input data in terms of pressure drop data for the components have also been used in the model. The pressure drop data for some of the components are already shown in the section 11 of this report.

For steady state simulations, the model has been run at a constant speed and loading condition. The model has been run at full load condition, and all the parameters have been evaluated in this particular. The thermostat is in closed condition, and based on the temperature from the engine, the valve of the thermostat lifts, thereby closing the bypass port. But the actual functioning of the thermostat can be seen in the results from transient simulations result. All the calculations have been made at an ambient temperature of 25^0 C.

The steady simulations have been carried at the following conditions as shown in Table 2. The torque and speed values shown in the Table 2 below are the full load condition performance results taken from the combustion calculations for the HDEP16 DST, Euro 6 engine

Table 2: Load Points and Ambient condition data

Speed	Torque	Air compressor heat load (kW)	HT pump drive ratio	LT pump drive ratio	Ambient Temperature (deg C)	Ambient Pressure (bar)
rpm	N-m					
1900	Tq1	6.7	1.89	2.98	25	1.01325
1800	1.085*Tq1					
1700	1.485*Tq1					
1600	1.220*Tq1					
1500	1.259*Tq1					
1400	1.315*Tq1					
1300	1.315*Tq1					
1200	1.315*Tq1					
1100	1.315*Tq1					
1000	1.315*Tq1					
950	1.315*Tq1					

13 Implementation

13.1 AMESim

All the necessary input data (from both supplier tests and in-house tests) relevant for all the components for example the pressure drop data, heat load data, engine performance data were collected. These data were modified according to the input format of the AMESim elements with proper units. As far as possible, the latest test data have been used, but for certain components, the latest data were not available as they were still under development and hence the data of the component's previous version were taken.

All the component elements of AMESim were arranged according to the architecture with correct routings of pipes and hoses. Most importantly, the geometrical dimensions of all the components were measured accurately as far as possible from CAD modules, as the coolant pressure drop, flow and Reynolds number are greatly dependent on the dimensions.

For accurately modelling the forced convective heat transfer coefficient across the engine cylinder head, cylinder block, EGR cooler and also to model the warm-up behaviour of the engine, the internal flow convective element and radial conductive exchange element respectively available in AMESim libraries were made use of. But because of the complex geometrical features of the above mentioned components, certain approximations were made, but overall, the geometrical features were kept as close as possible to the reality.

The transient simulations were performed for Borås-Landvetter cycle. The inputs in terms of the speed and torque changes were made to affect the heat load applied to the engine cylinder head and the block. The transient responses to the Transmission oil cooler, Oil cooler were not taken into account because of the usage of oil as a working fluid. The models of these oil system components are under development and are not covered in the scope of this thesis, and hence the heat loads on these components are assumed to be constant.

The input heat load, charge air mass flow, EGR mass flow etc. were applied to all the components using the maps of above mentioned parameters. All the maps are driven by speed and load. For transient simulation, the changes in engine speed, vehicle speed and engine torque according to Borås-landvetter cycle were taken in the form of a map and were made to affect the changes in EGR mass flow, EGR temp, CAC mass flow etc. as shown in Figure 8 below.

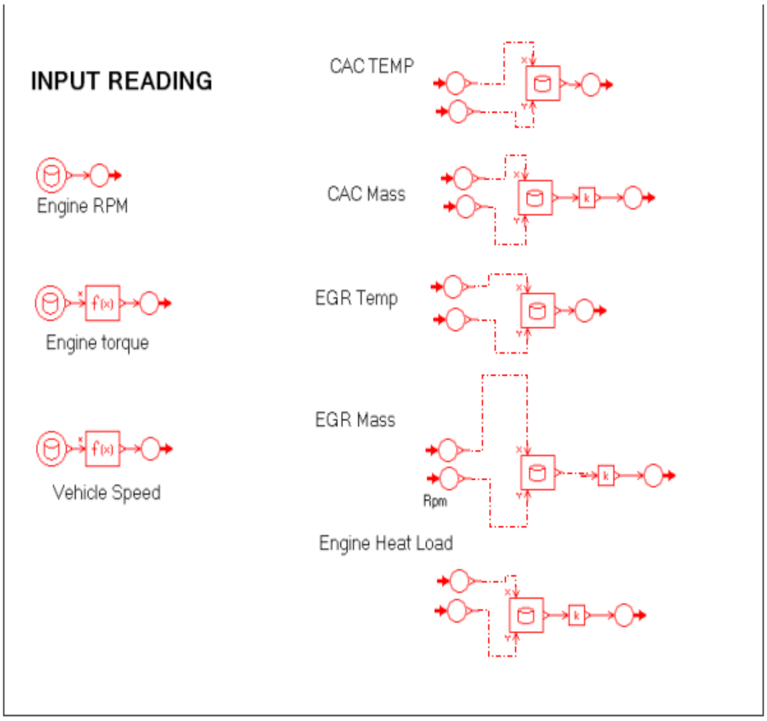
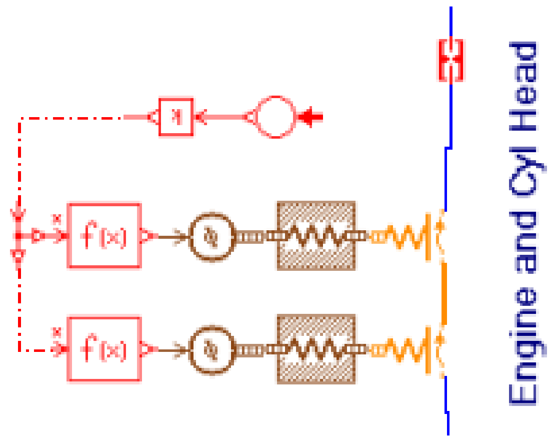


Figure 8: Input data as seen in AMESim

The model to determine the heat transfer coefficient for engine cylinder head and cylinder block is implemented as below.



The total heat load is split into 65% to the cylinder head and 35% to the cylinder block. This split was arrived at from the combustion calculations. To determine the heat transfer coefficient, it is important to determine the Reynolds number of the coolant flow across the head and block accurately. The characteristic dimension for the cylinder head and block has been determined by taking out the cross sections in slices across the whole length of the head (both upper and lower deck) and block in the direction of the flow and integrating all the slices and averaging out the integral value. The actual geometry of the head and block is shown in Figure 9 below.

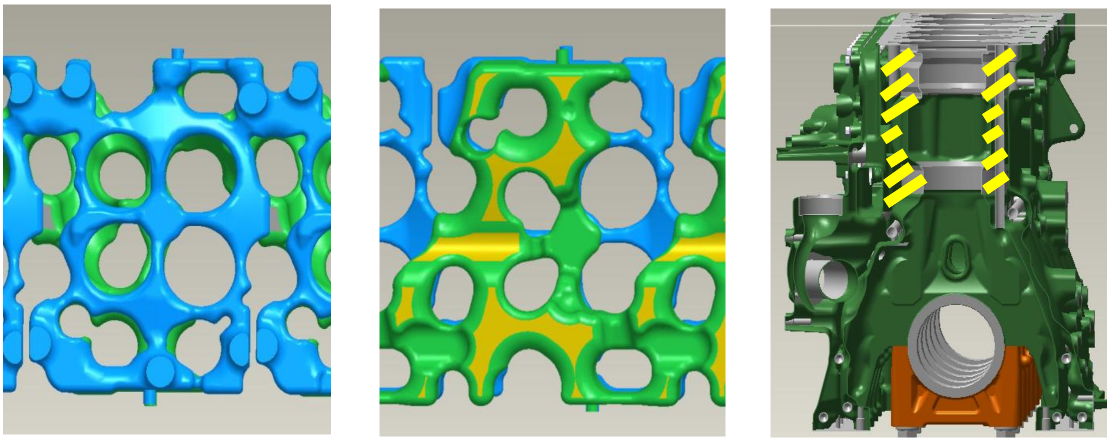


Figure 9: Cylinder Head and Block coolant flow geometry

The Nusselt number correlation in the internal flow convection element of AMESim was changed from default Dittus-Boelter equation to the modified convective heat transfer coefficient as shown below.

$$Nu = (0.023 * Re^{0.8} * Pr^{0.4}) * \left(\left[1 + 23.99\, Re^{-0.23} * \left(\frac{x}{D_h}\right)^{n}\right]\right) * \left\{\frac{1}{\left[1-\left(\frac{x_0}{x}\right)^{\frac{9}{10}}\right]^{\frac{1}{9}}}\right\} * \left[0.091 * \left(\frac{\varepsilon}{D_h}\right)^{-0.125} * Re^{0.363\left(\varepsilon/D_h\right)^{0.1}}\right] * \left(\frac{\mu_{bulk}}{\mu_{wall}}\right)^{0.14}$$

The hydraulic diameter or the characteristic length was changed in the above equation depending on the cross sections of the cylinder head and the cylinder block.

The fan model built as shown in Figure 10 is a simple fan model taking into account only the temperature at the inlet of thermostat and can basically work on 3 different temperature regimes – temp < 96 deg C , 96 <= temp <99 and temp > = 99 deg C.

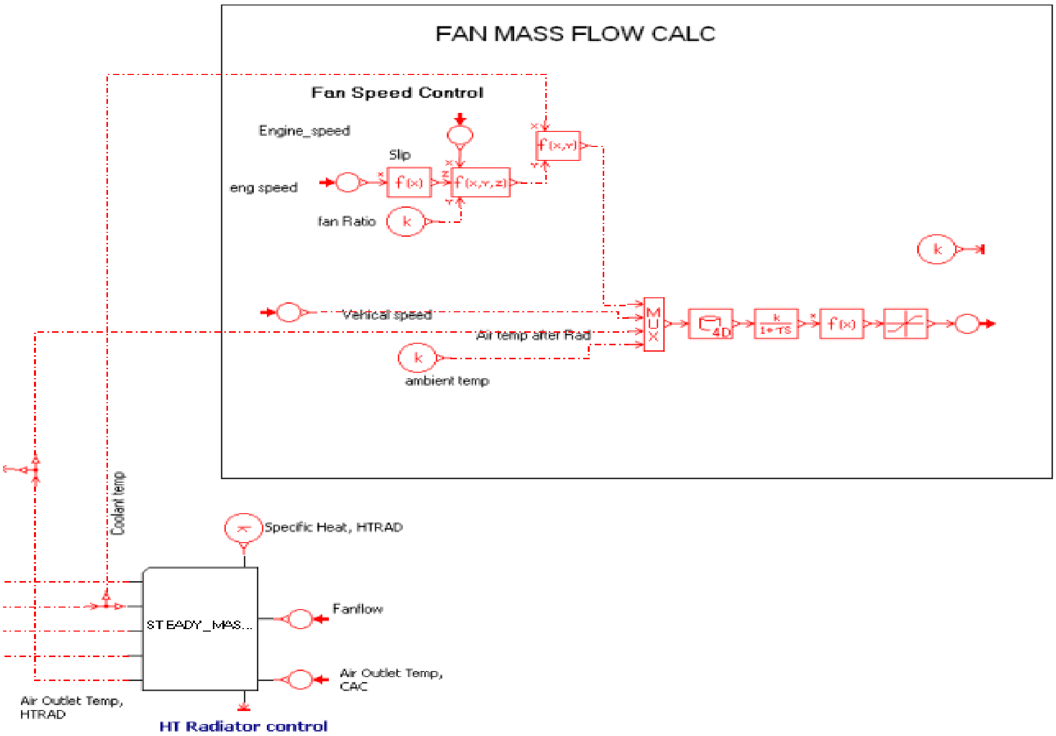
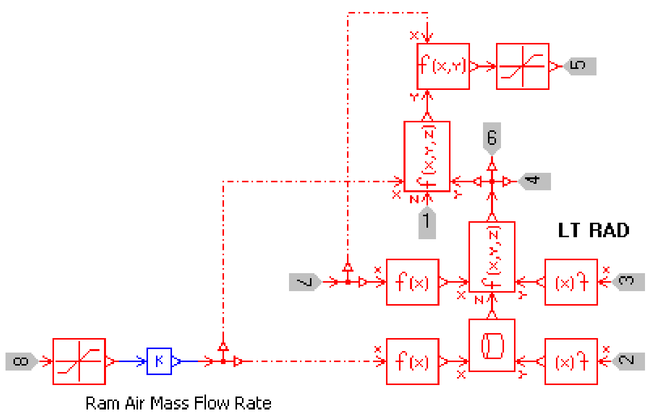


Figure 10: Fan model as implemented in AMESim

The fan model has been built taking into account the fan slip, the fan speed ratio, temperature of the coolant at the inlet of the thermostat, engine speed etc. The fan mass flow rate is calculated using a map which has 4 inputs namely – Fan speed taking into account belt slip and fan drive ratio, Vehicle speed, air temperature after radiator and ambient temperature.

The radiator heat transfer logic is built using the below model. The same model has been used for calculating the heat transfer for the main radiator as well.



The inputs 2 and 8 which are the mass flow rate of ram air and mass flow rate of coolant are fed into the performance map of the front radiator which gives the heat load data. But the performance map is generated for the radiator at a temperature difference (between coolant and ram air temp) of 50 deg Celsius and hence the heat load values generated from the map needs to be adjusted for the real temperature difference occurring in the model. Similar adjustments have been made for calculating the heat load for front radiator, EGR cooler, charge air cooler as all these components have been tested at different delta T values.

13.2 GT Cool

The major motivating factor for developing the cooling system model in GT Cool was on one hand to compare two different tools viz a viz AMESim and GT Cool and on the other hand to couple the model developed in GT Cool with that of the combustion system model. In Volvo Powertrain, GT Power is the main tool used to develop combustion system model and it is logical to couple two different systems developed in the same tool to understand how they interact with each other because of the close relationship between these two subsystems.

In addition to the basic inputs collected for AMESim, GT Cool required some additional inputs in terms of geometric details of the radiator, EGR cooler, Charge air cooler etc. for calculating the heat load. The logic for calculating the fan mass flow was implemented as shown Figure 11 below. Here also, a fan map was used to calculate the fan mass flow rate.

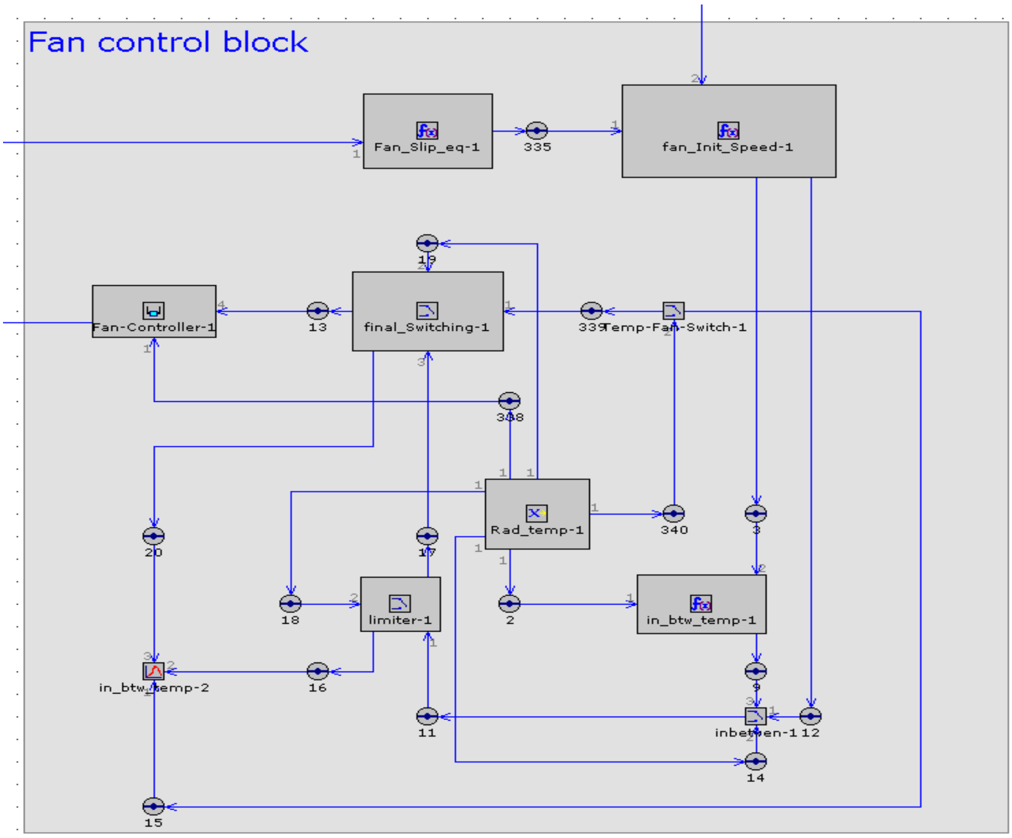
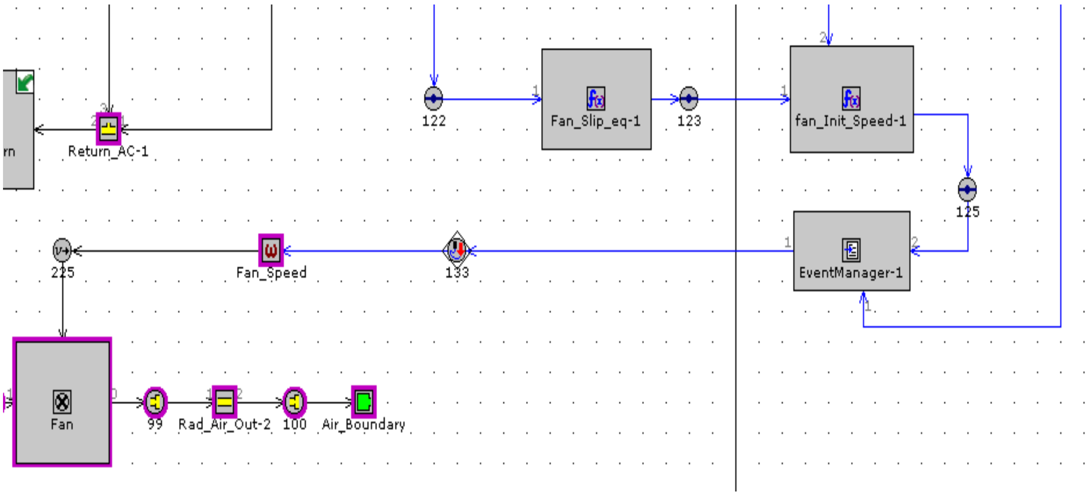


Figure 11: Fan model as implemented in GT Cool

The same logic as used in AMESim has been implemented in GTCool with the help of elements of GT Cool. The fan speed regimes are controlled using a Switch and a PID controller (used to quickly brings the system to a steady state value), which basically controls the speed of the fan based on the temperature at the inlet of the thermostat. The suitable gain values for PID controller were calculated based on trial and error method. As an alternative, another element called as the ‘Event Manager’ was used and the same is shown below.



Event Manager basically evaluates the coolant temperature and based on that, controls the fan speed. This way of implementation lead to a lot of fluctuations as the whole system reacts a bit slowly from the Event Manager. The front radiator, charge air cooler, main radiator and fan cluster is arranged in the following manner as per the architecture shown in Figure 12 below.

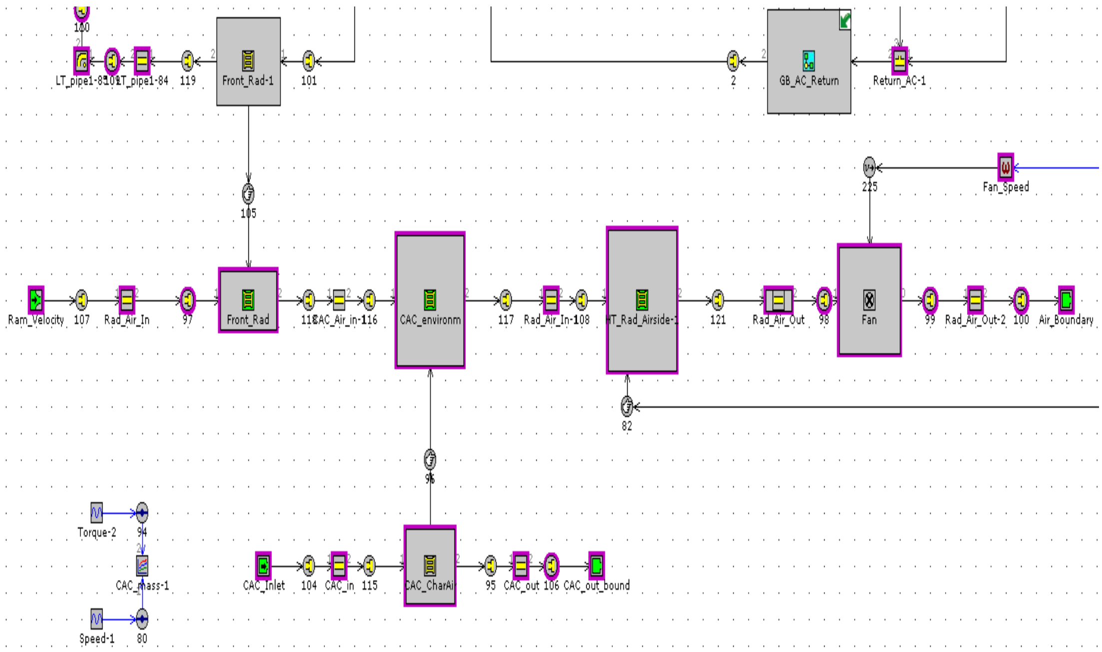


Figure 12: Radiator arrangement in GT Cool

The end environments for charge air cooler, radiator, EGR cooler have been setup using the same maps as used in AMESim using the ‘Signal Generator’ and ‘lookup 2D’ elements.

Here also all the maps are driven using inputs from engine speed, vehicle speed and engine torque and for transient simulations, instead of giving constant values, these signal generators are given transient speed and load maps.

In the transient simulation and as well as the steady simulation the heat load was used as an input map as shown below in the Figure 13. The composite heat transfer correlation which was used in AMESim has been used in GT Cool as well. But as shown in Figure 13 the implementation way is a bit different wherein, the heat transfer coefficient has been calculated separately using the composite equation and this heat transfer coefficient was made to affect the heat transfer on the pipes. This is explained in details below with corresponding picture for easy understanding.

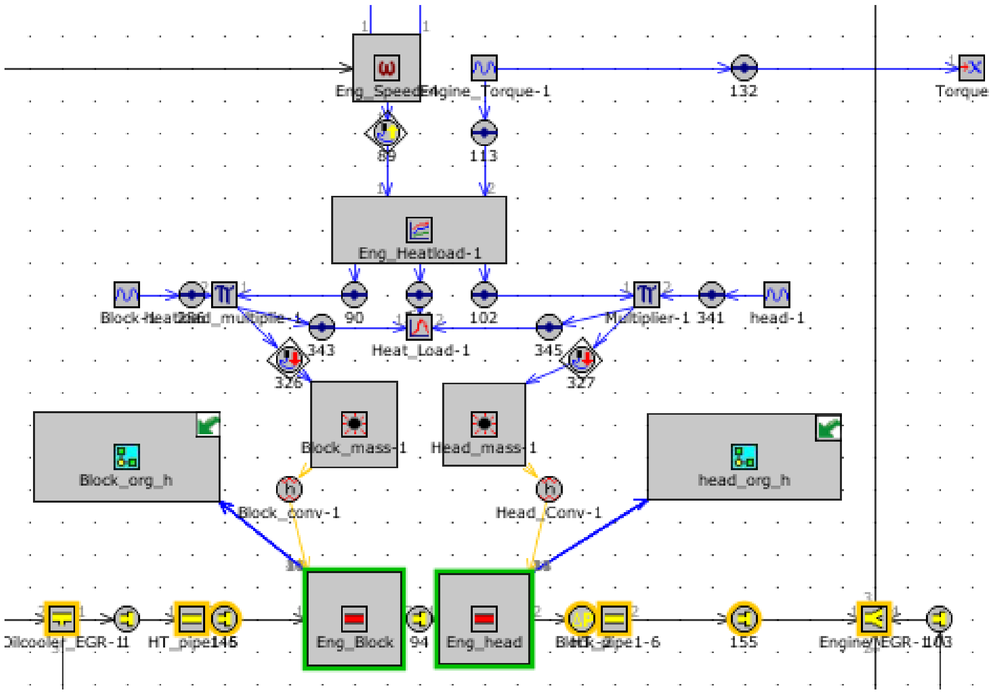
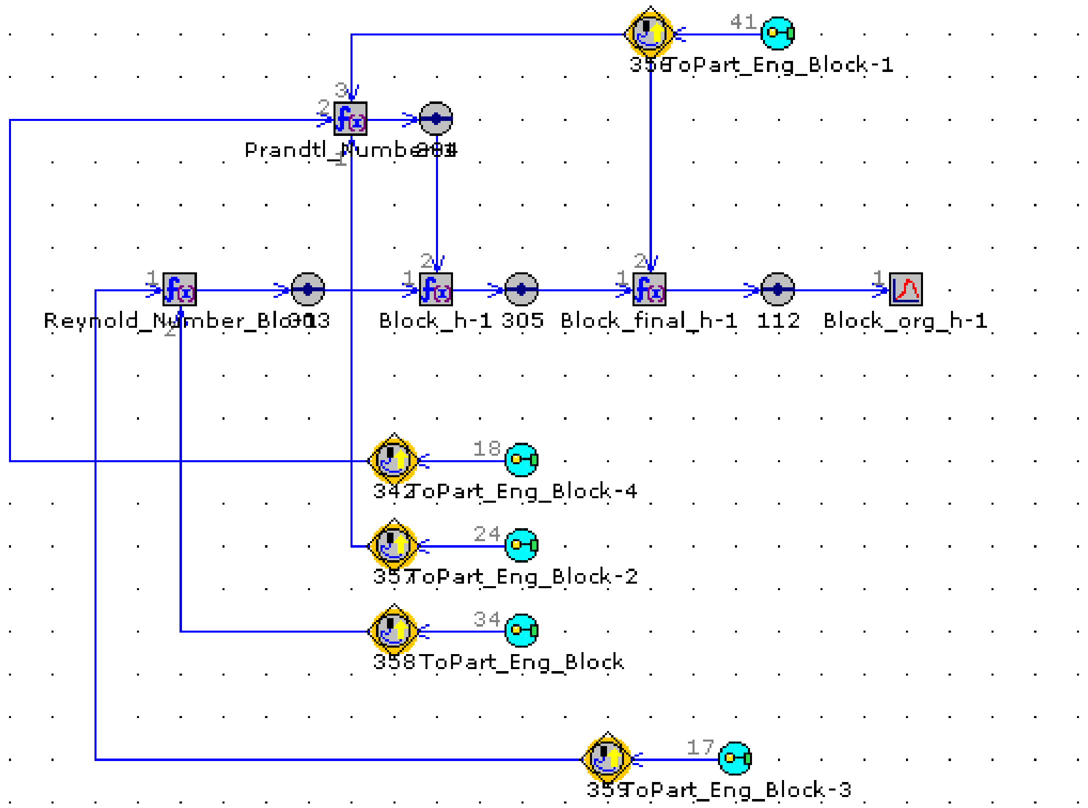


Figure 13: Coposite correlation implementation for Engine warm up behaviour representation

The heat transfer value calculated separately using the composite equation was divided by actual heat transfer value calculated by the tool to arrive at a factor. This factor was used as a multiplication factor and was made to affect the heat load on the pipes which in turn is used to represent the flow through the engine block and head. The surface area corresponding to the geometries of the block and cylinder head were carefully made to represent in the pipe elements used. The thermal masses representing the head and block were also used to represent the warm-up behaviour of the engine. There was however one difference between how it is implemented in GT Cool compared to AMESim. In AMESim, the warm-up of engine cylinder head is measured even within the thickness of the cylinder head walls. In other words, the conduction phenomenon within the walls is measured at 3 places. But in GT Cool, this phenomenon is not represented and the warm-up behaviour is represented as a bulk phenomenon and conduction within the wall is not represented. In other words, the heat up pattern within the walls is not represented in GT Cool.

With regards to calculating the forced convective heat transfer, it was a straight process to implement in AMESim, where the modelling element allows the use of customised heat transfer correlations apart from standard Dittus-Boelter equation.

In GT Cool, the elements do not allow the use of customised correlation. If, one wants to implement customised correlations, then it can be done by writing a code in a user subroutine which is a bit complicated. Another way, is to use a set of controls and sensors elements along with some math template elements and write the correlation, and then based on the result try to impose the heat transfer as a multiplier on the pipe elements used representing the engine. In this Thesis, the latter approach is used as shown below.



14 Steady State Simulation Results

Since, HDEP16 DST is an on-going project a lot of changes in terms of architecture is being made. But with respect to this thesis, all simulations were made using the architecture shown in section 11.

Also, a lot of components are in the development stage but the results shown below are computed based on the latest input data available, at the time of running the model.

The flow and pressure drop data for the critical components of the cooling system is shown and possible reasons for variations of results have been highlighted. With regards to thermal results, the hard points corresponding to the actual input data available from combustion simulations should be looked into in detail for comparison between modelling tools and other points shouldn't be used for comparative study as these points are not correct because of interpolation method used to generate part load input data. These part load results should be used for analysing the trend of a particular performance result.

It is also to be noted that, all the pressure drop values shown below are pressure drop across the components excluding the hoses/pipes connecting those components. It is pointed out that, the model in AMESim and GT Cool is same in all aspects and all the geometrical dimensions for pipes/ hoses and all the components have been kept the same.

14.1 Coolant Pump

The pressure rise curve for the coolant pump is shown below in Figure 14. The curve below shows the pressure rise, coolant flow vs. the engine speed.

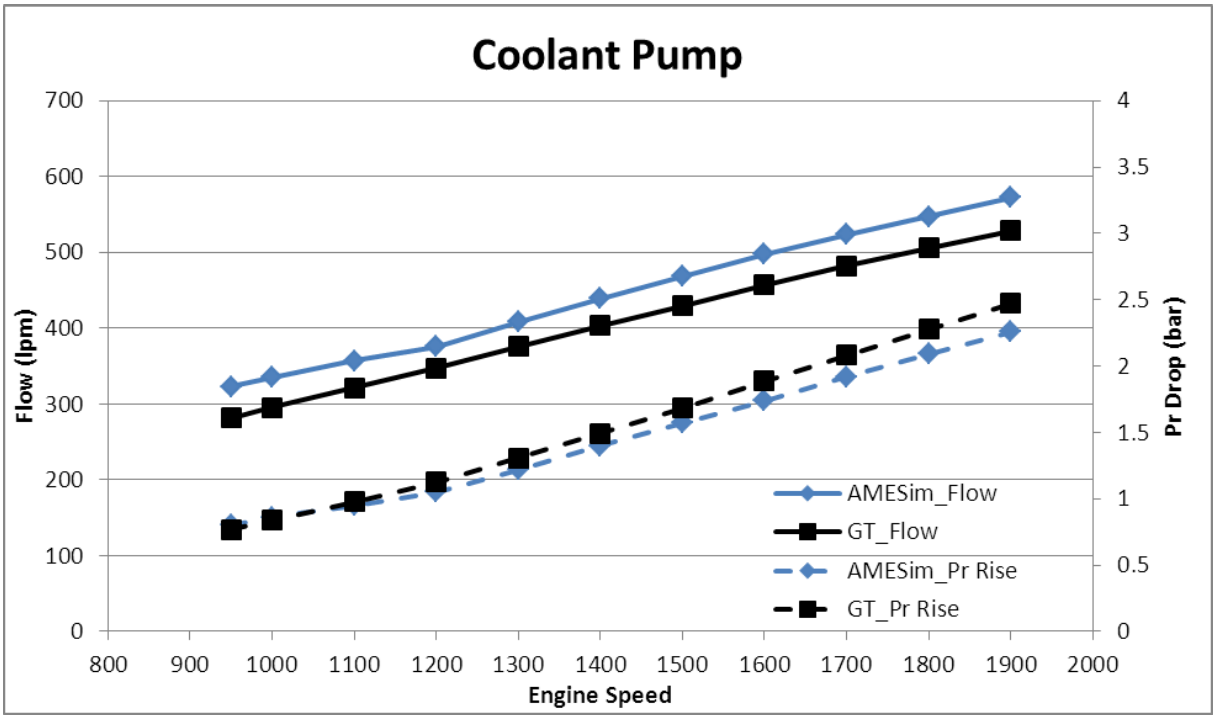


Figure 14: Pressure Rise, Flow Vs Engine speed

It can be noted that there is a big difference between the flow results between AMESim and GT Cool. On the other hand the difference in pressure rise between the two simulation tools suggests a peculiar behaviour.

It can be seen that, at lower rpm's the difference is very small, but as the speed increases, the difference in pressure drop increases gradually. Figure 15 shows how the pressure rise varies with flow.

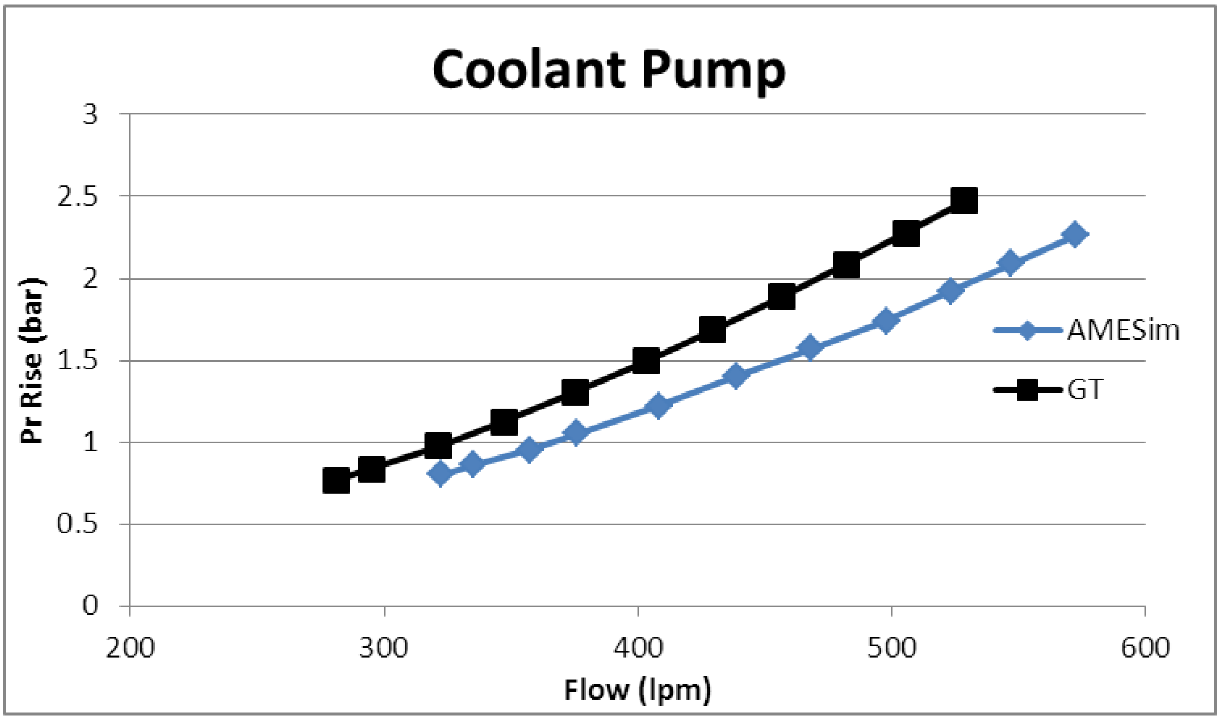


Figure 15: Pressure rise Vs Flow

From the graph above one can conclude that the coolant flow shown by GT Cool is lower for the same pressure rise as compared to AMEsim because of the difference in the inlet pressure to the coolant pump.

An investigation into why there is a difference in pressure between the inlet and outlet of the pump in the two modelling tools was also done. In this study, the major flow components of the cooling system, viz Oil cooler, Engine, Retarder, Thermostat were studied for pressure drop (study included just the component pressure drop excluding the hoses/pipes). Rest of the components in the model were ignored for this study as coolant flow through these components are very low compared to components listed above. The aim behind this particular study was to understand whether the pipe element used in GT Cool, uses a different correlation and whether the Reynolds number defining the transition between laminar and turbulent region were defined in a different way.

As stated earlier, the components chosen for study uses a pressure drop table and the tool doesn't calculate the pressure drop on its own for these components and instead uses the table. But in order to determine how the tool calculates the pressure drop for pipe/hoses this study was performed. The result below basically depicts the following. The result from this investigation is shown in Figure 16.

Pr_drop hoses/pipes = Pr Rise Pump – Pr drop oil cooler – Pr drop Engine

– Pr Drop Retarder –Pr drop Thermostat

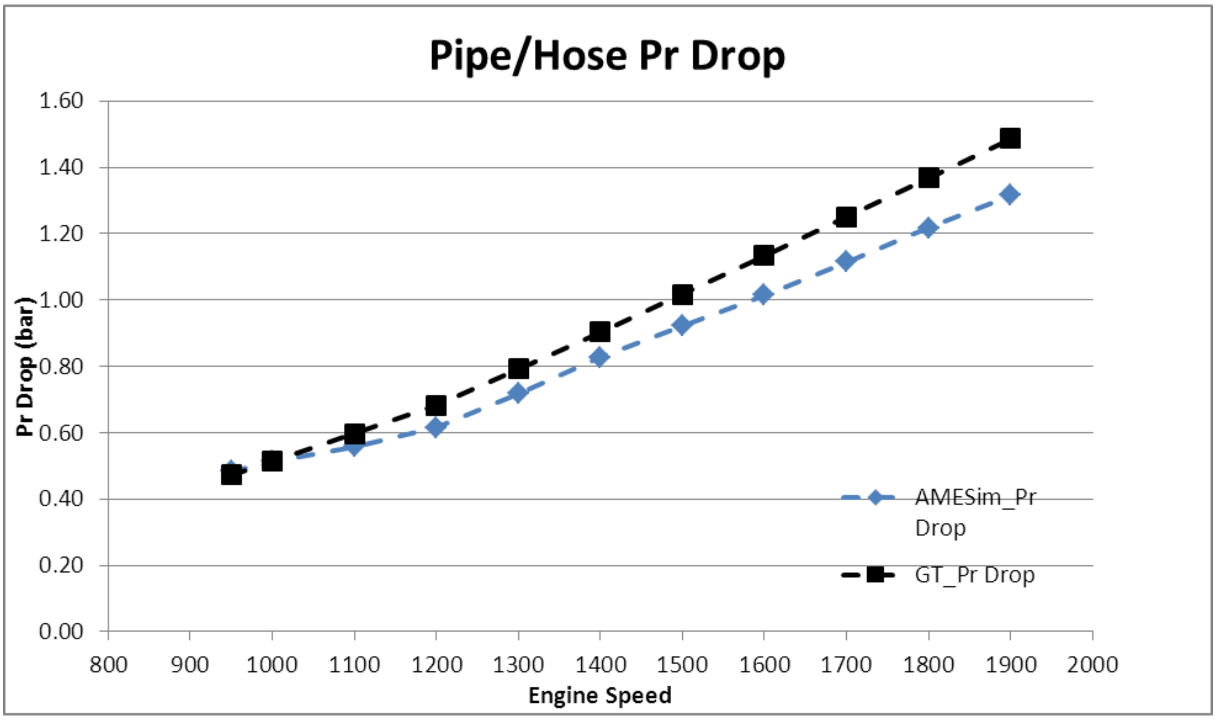


Figure 16: Pressure drop in pipes/hoses

The difference in pressure drop for pipe/hoses calculated by GT Cool increases as the speed increases. The difference in pressure drop value calculated by GT Cool is around 10-12 % higher than AMESim and this result in a lower output flow rate from the pump, as the pump's output flow rate is calculated from a pump map which basically gives the output flow as a function of coolant pressure rise. The above result clearly demonstrates that GT Cool defines the Reynolds number for transition from laminar to turbulent differently compared to AMESim.

The pressure drop depends largely on the Reynolds number. The Reynolds number is used to determine the friction coefficient of the pipe and the pressure drop is a function of this friction coefficient.

The result of this change in pressure drop and in turn coolant flow rate from the pump has a cascading effect on all the components downstream of the cooling system modelled and hence all the components will show varying percentage of difference in flow and pressure drop between the two modelling tools based on the geometry of the components and their connecting pipes/hoses.

At this stage it will be very difficult to out rightly point out which tool is showing the correct behaviour and hence the results from two modelling tools needs to be verified by conducting physical tests on the engine. It is however pointed out that, the physical tests on the engine couldn't be carried out because of unavailability of various parts and inability to assemble the engine on time for testing.

14.2 Heat Load

The graph in Figure 17 basically shows the normalised coolant heat load from the engine. The engine heat load is given as a map which is a function of engine speed and engine torque. The error in how the heat load is output to the coolant from the map by the two modelling tools is shown below.

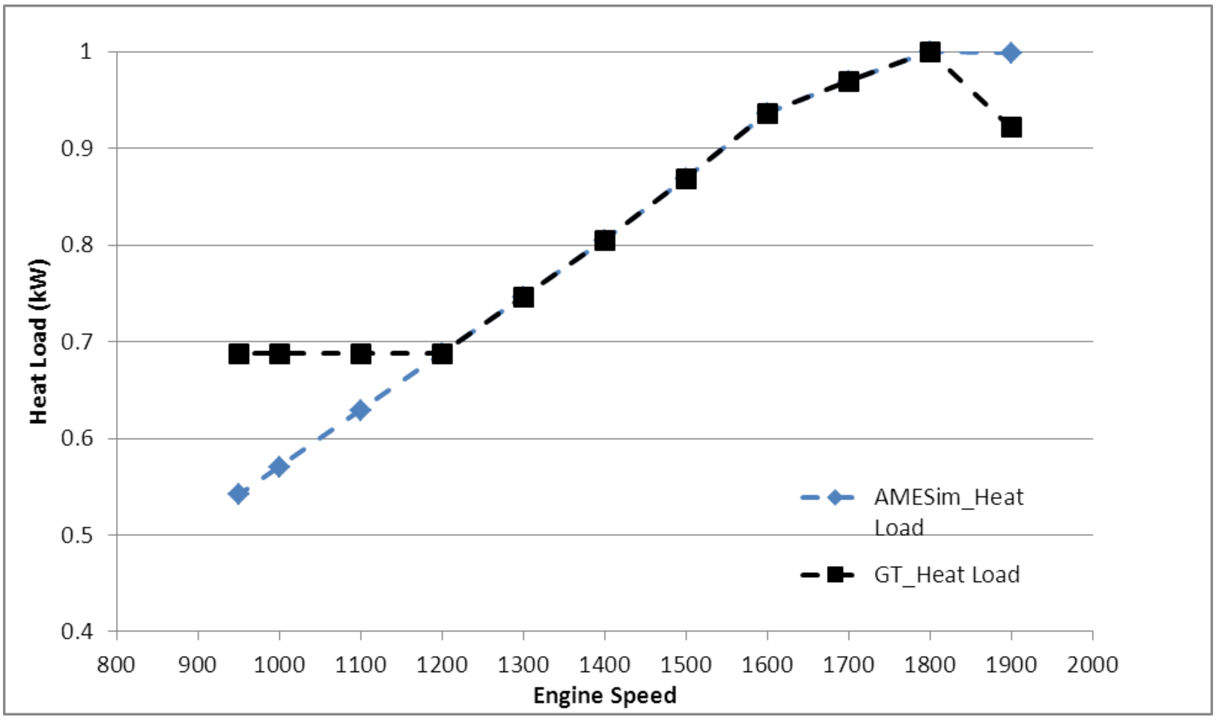


Figure 17: Engine heat load (Normalised)

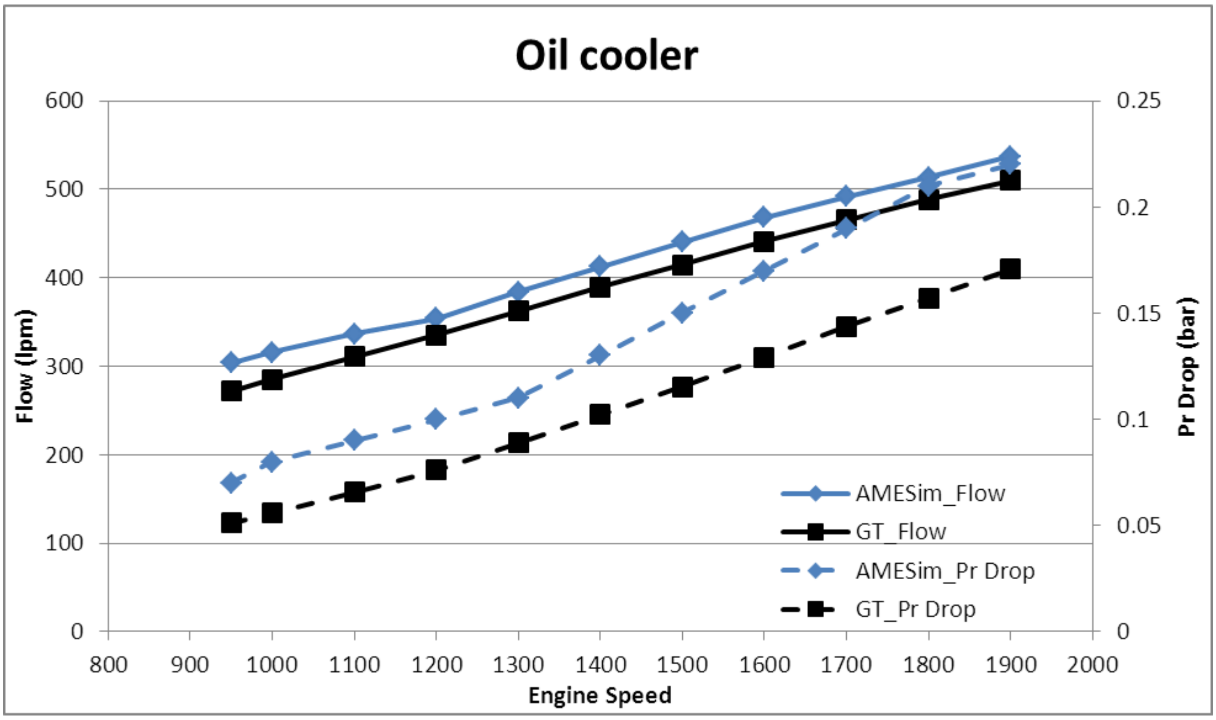
If one look at the data shown in the section 12 of this report, it can be seen that the heat load corresponding to the hard points (marked in bold letters) given in the table is what is output as the coolant heat load by both the tools. But for speeds outside the hard points (1200 and 1800 rpm), both the tools use different extrapolating techniques and hence there is a difference in the result as shown above in the graph. In GT Cool, it requires actual minimum and maximum values to be mentioned as boundary condition in the map. If these values are not available, then the map reads the data corresponding to the last boundary point value mentioned in the map for all input values outside these boundary points. This is very evident for GT Cool results for speeds from 950 to 1200 rpm, where the heat load is constant, which is nothing but the heat load corresponding to the hard point at speed of 1200 rpm. However, after 1800 rpm, this behaviour is not seen and there seems to be a different behaviour. But after reading the GT Cool documentation it was ascertained that the extrapolation of values is not possible in GT Cool and the values shown for speed below 1200 is correct.

This way of map reading outside the speed boundary of 1200 rpm and 1800 rpm leads to a significant divergence of overall results in transient simulation as the engine operating on a Borås-Landvetter cycle is predominantly in the range of 800 to 1270 with short bursts of speeds going around 1500 rpm. Added to this is the low flow from coolant pump in GT Cool on account of increased difference in pressure between inlet and outlet of the pump. The factors amounting from heat load and coolant flow results in a big difference in results of GT Cool simulation and AMESim results particularly for transient simulations.

From the Figure 17 it is very evident that particularly with respect to heat load outside the boundary condition GT Cool does not give a good result, and the decreasing heat load trend shown by AMESim outside the 1200-1800 rpm boundary seems to be correct.

At this point it is also important to point out that, since the heat load of various components like EGR cooler, Radiator etc. are used as a map which is a function of mass flow rate of two fluids (coolant to EGR or Coolant to ambient air), the heat load output from component in both the tools will be different because of the change in coolant flow rate, the reasons of which are explained in section 14.1.

14.3 Oil Cooler



The lower coolant flow rate from the coolant pump shown by GT Cool results in a lower coolant flow rate across oil cooler. As shown in the graph above the flow through oil cooler is lower resulting in a lower pressure drop as compared to the result in AMESim.

14.4 Engine

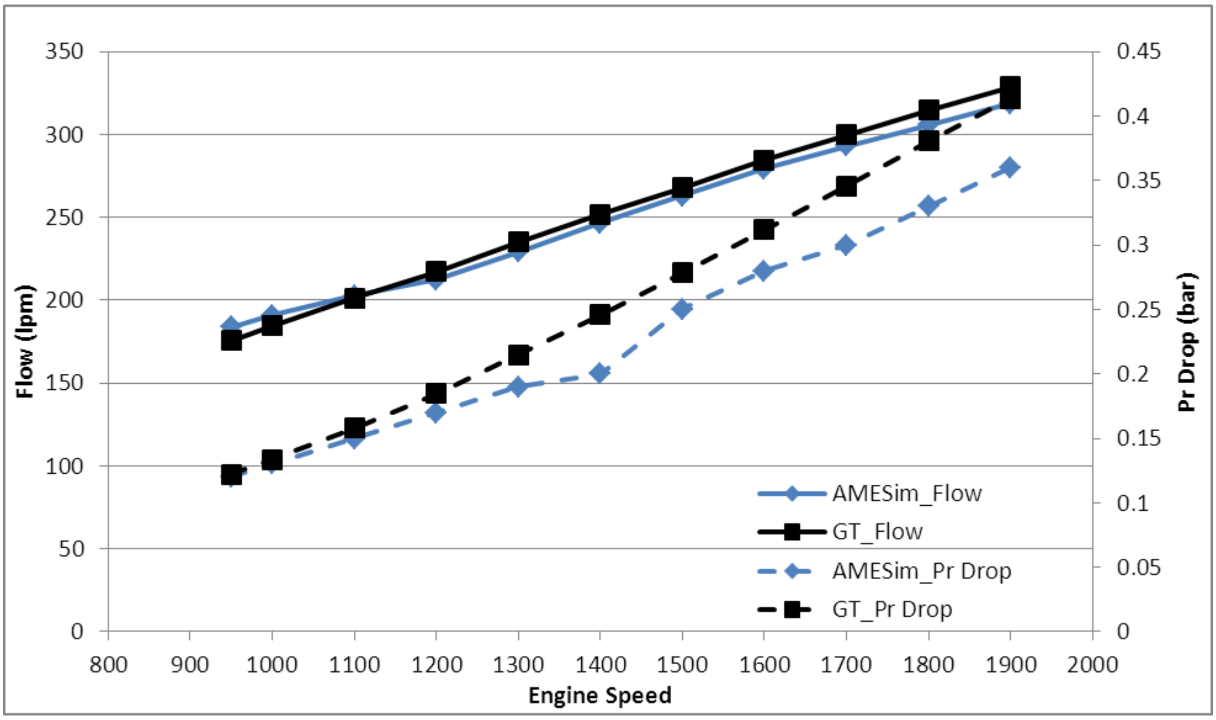
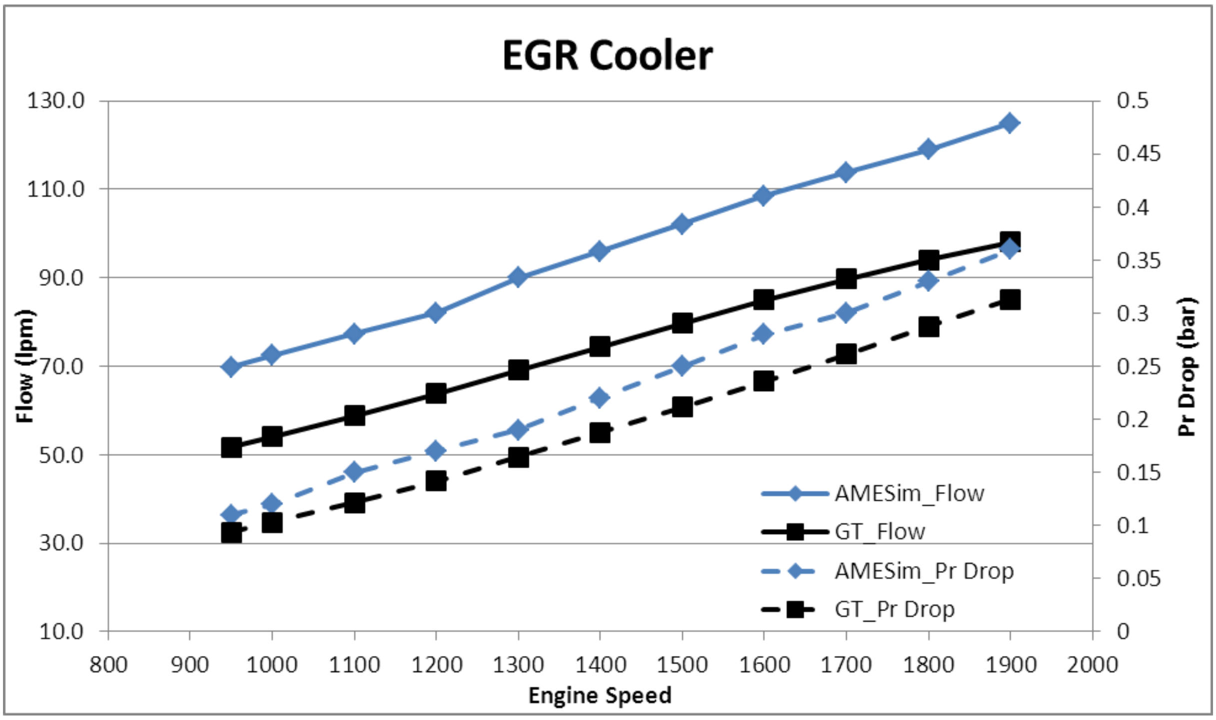


Figure 18: Pressure drop across engine

Although the pressure drop across the engine should have been the same for both the tools, as tool just reads the coponent pressure drop map, but the change as shown in the Figure 18 results from the change in flow resulting from pressure drop across the pipes/hoses before the engine as was explained in section 14.1

14.5 EGR Cooler



The biggest difference in flow occurs majorly in two components viz EGR cooler and CAC. The flow parameters for CAC can be seen in the section 14.6. The only reason which can be attributed to this is the parallel arrangement of EGR cooler and the Engine. If one looks at the engine result, the coolant flow through the engine is comparatively more in GT Cool than in AMESim. This basically suggests that, there is a low resistance to flow in the branch going to Engine block and head compared to the branch going to EGR cooler.

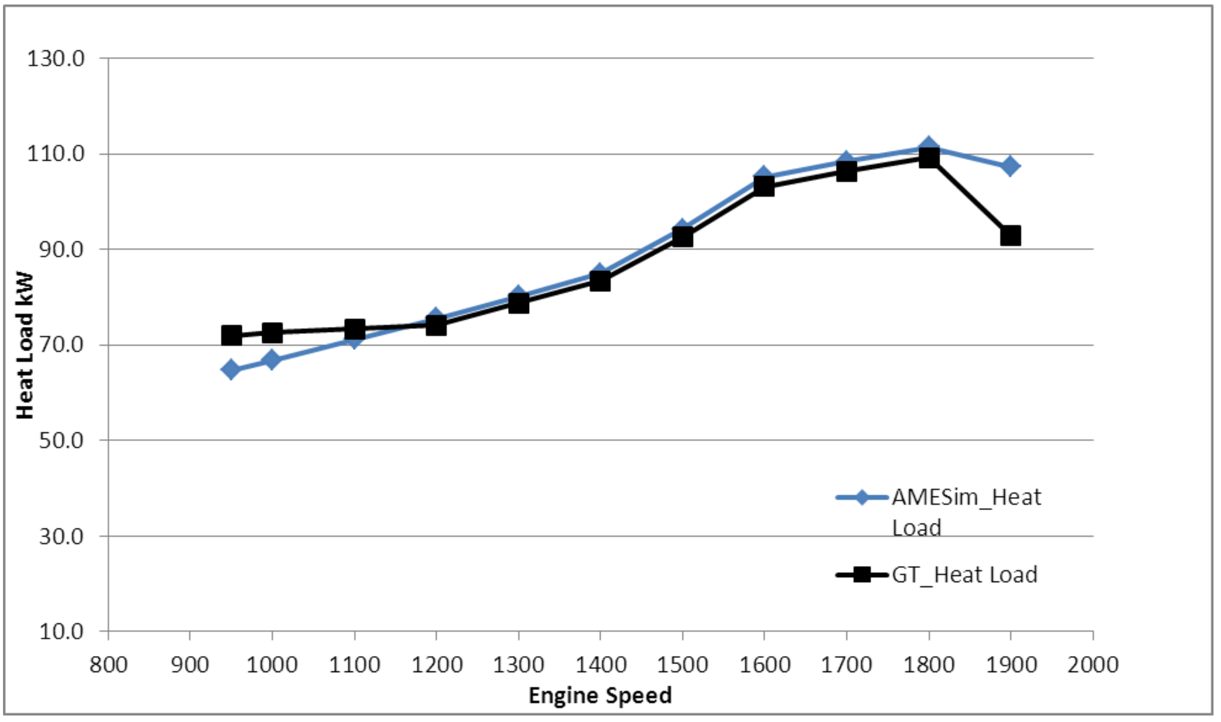


Figure 19: EGR Cooler Heat load

From the graph shown in Figure 19 it can be clearly seen that from 1200 rpm to 1800 rpm engine speed, Both the tools calculate the heat load from EGR cooler in the same manner. It is pointed out that EGR cooler heat load is calculated based on the component heat load map which is a function of temperature difference between both the fluids and fluid mass flow rate. Even though the heat load values shown by the two modeling tools are almost the same, because of the change in coolant flow and difference in coolant temperature before entering the EGR cooler, the outlet temperature of the EGR flow through the cooler will be different and this results in a change in the IMTD values shown in section 14.10.

14.6 Charge Air Cooler

The results shown in Figure 20 are for the CAC (charge air to coolant) and do not represent the charge air to ambient air cooler. It is also pointed out that, heat load for charge air to coolant CAC has not been modelled because of the unavailability of the CAC heat load data as the component was still under development when this thesis was being performed. Very similar results of EGR cooler can be seen for CAC below. Here also the arrangement of CAC is similar to that of EGR cooler. The CAC is in parallel to the oil cooler. The argument as proposed for EGR cooler is valid for CAC when we compare the CAC results to that of Oil cooler. The difference of flow in CAC and EGR cooler is very high, but in addition to the reason mentioned above, the overall lower flow through the coolant pump in results shown by GT Cool also has to be kept in mind which leads to this large divergence. The results from the two tools are shown below.

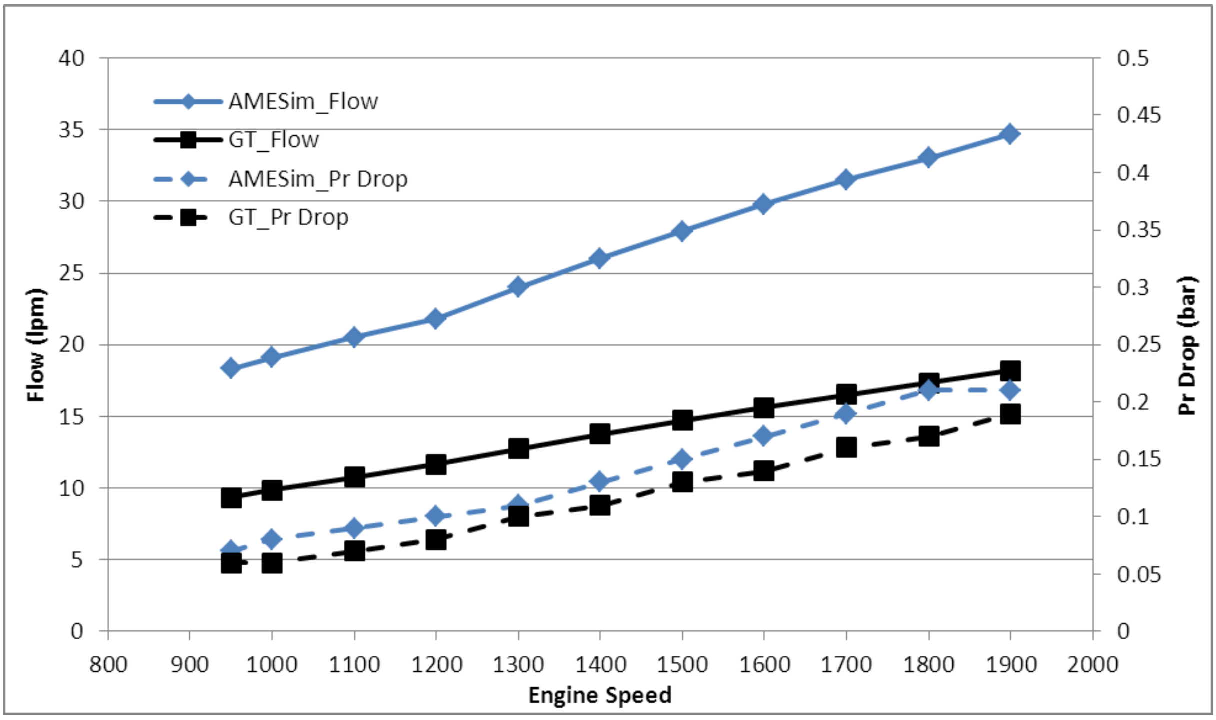


Figure 20: CAC pressure drop

Now, as pointed out in the limitations section, the air modelling is different in AMESim and GT Cool. Because of the lack of availability of time, air side model in AMESim has not been modelled taking into account the physics involved. A very simple strategy has been employed for modelling the air side in AMESim according to a simple map which gives the fan flow based on the speed of fan, ambient temperature, air temperature after the radiator and vehicle speed. But the modelling in GT Cool represents the real physics involved and is more accurate. Because of this variation there is difference in the heat load data values of the CAC (charge air to ambient air) as can be seen in Figure 21.

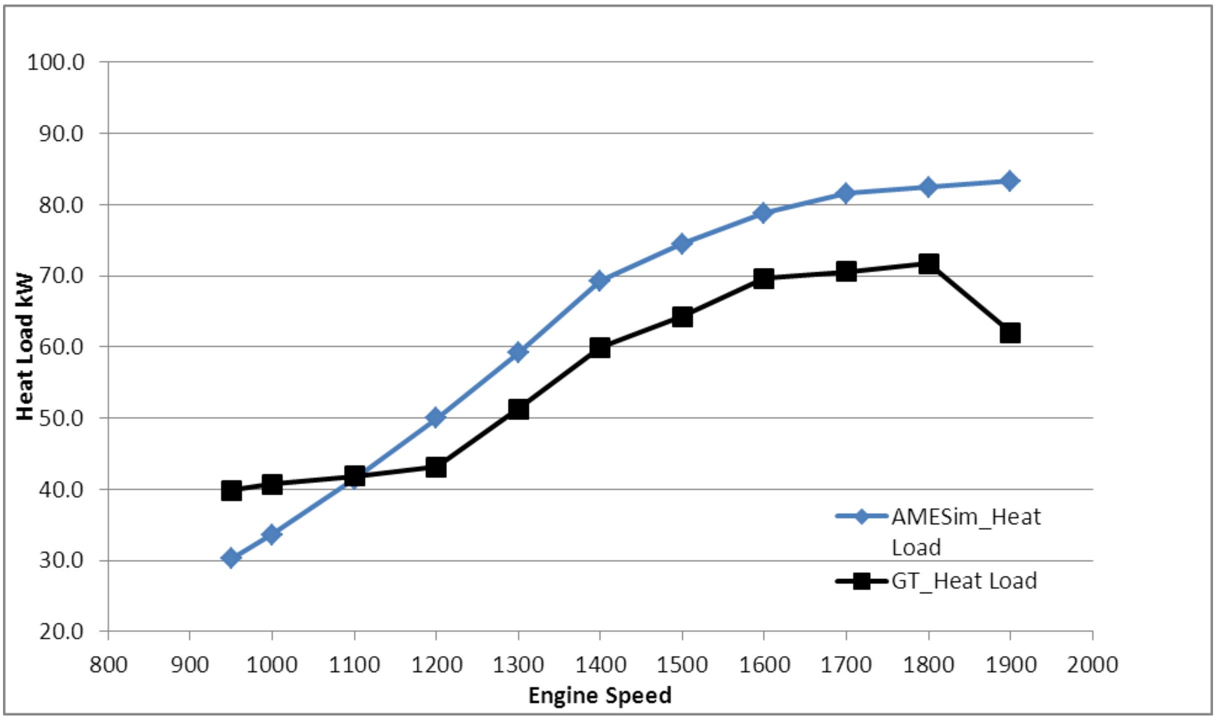
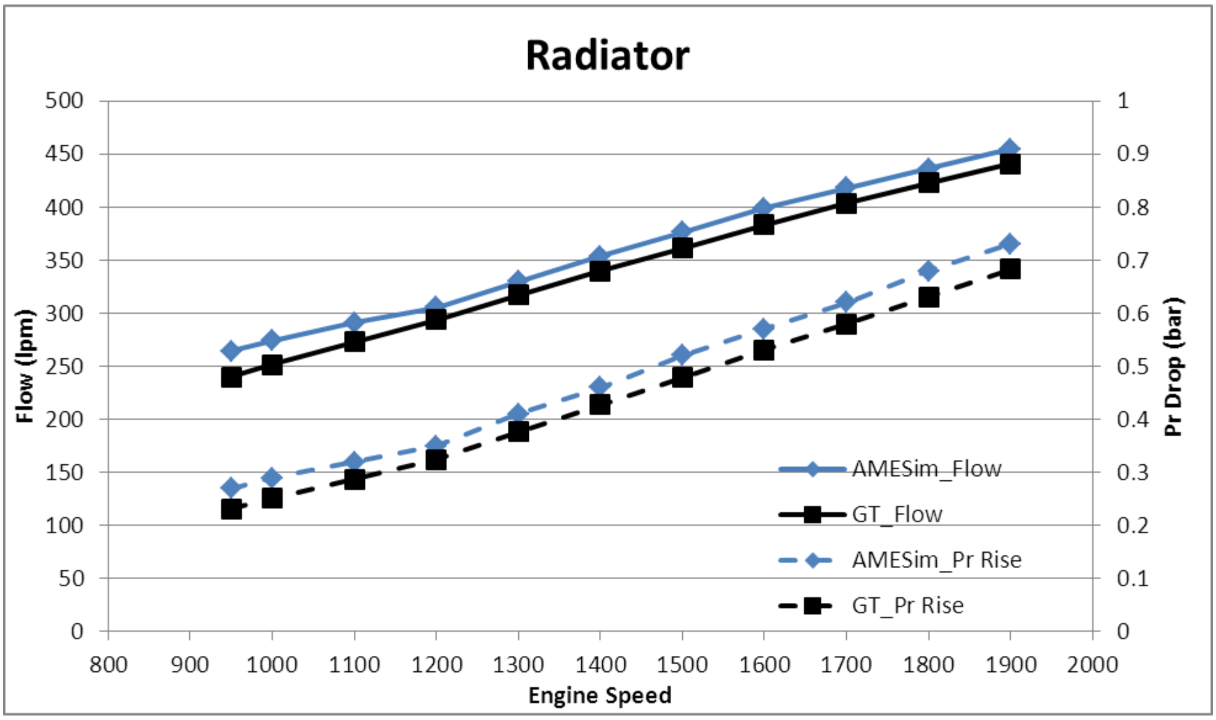


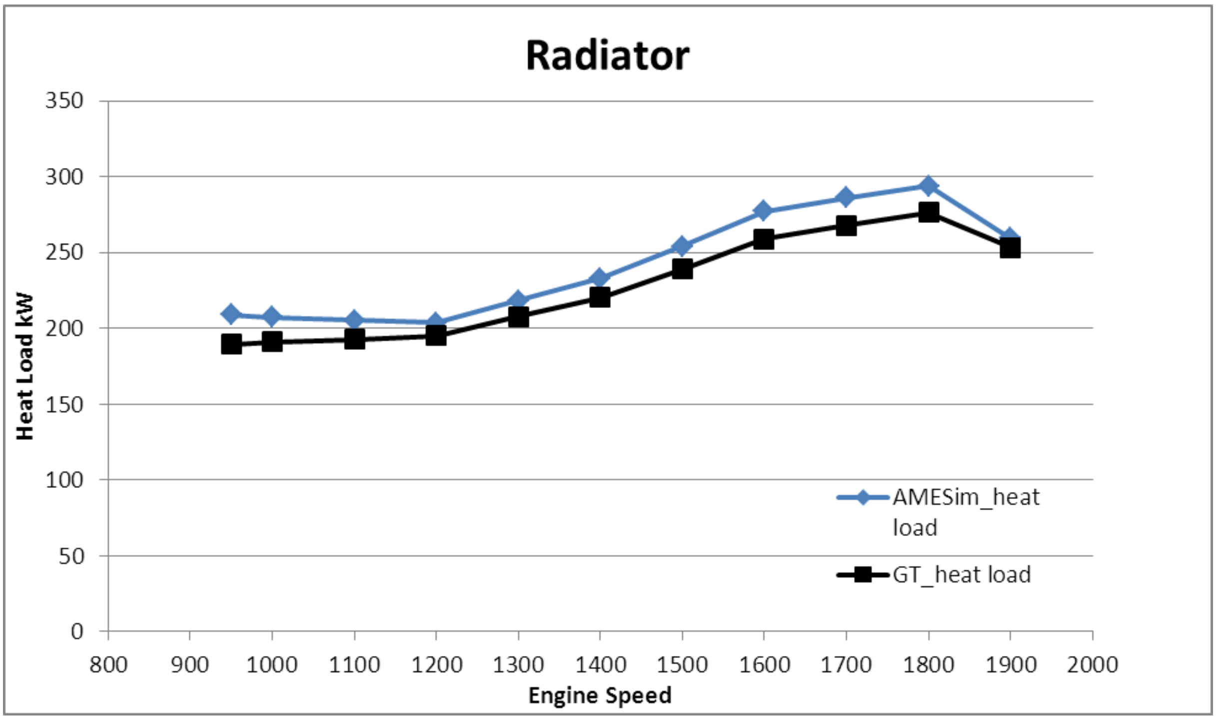
Figure 21: CAC heat load

The variation is the result of the fan flow, even though for heat load calculation same maps have been used.

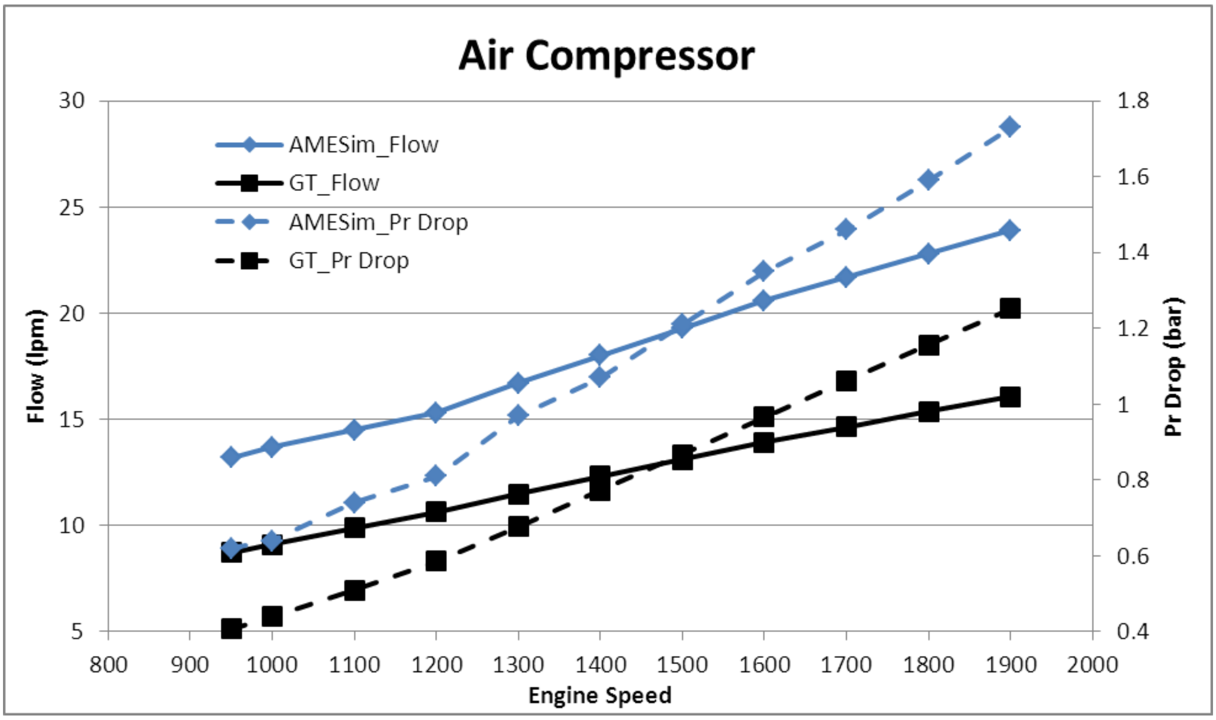
14.7 Radiator



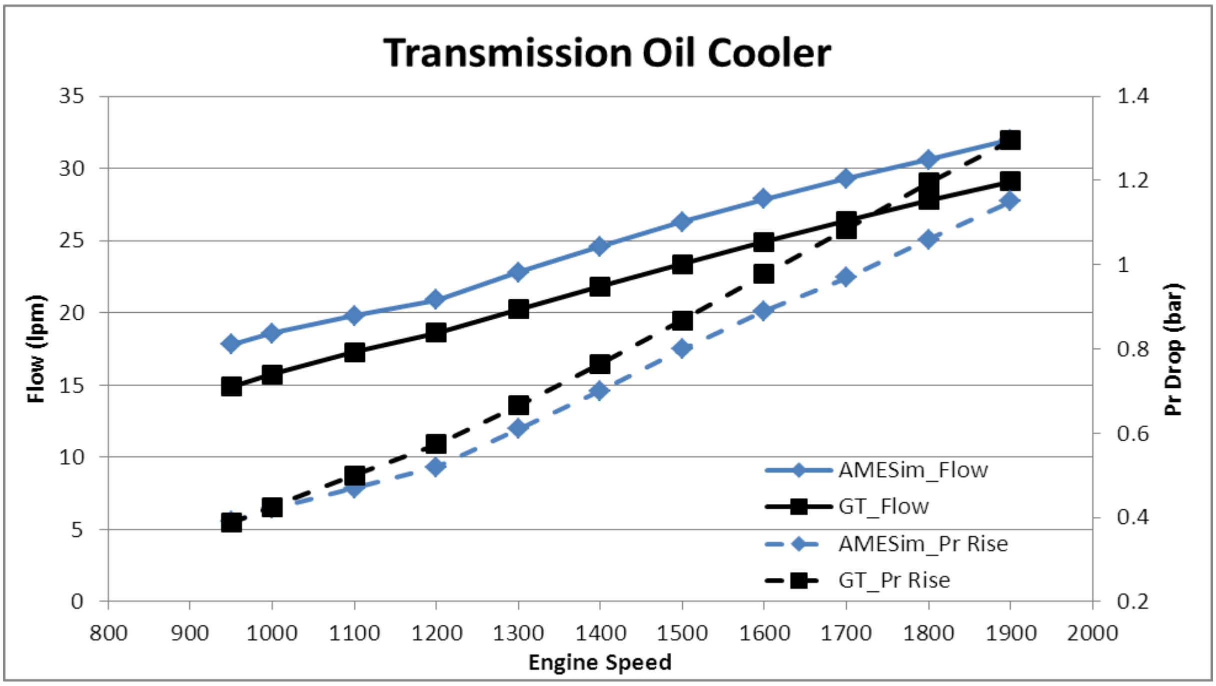
The flow and pressure drop values across the radiator shown by both the tools follows the same trend as shown in the results for the coolant pump. Similar results were found for retarder circuit which is in series to the radiator. But the results above have been shown for an open thermostat and the variation of flow through the radiator can be better seen in the transient cycle results. The heat load curve is shown below, where the above stated reason is valid for the change in the heat load values between the two tools.



14.8 Air Compressor and Transmission Oil Cooler

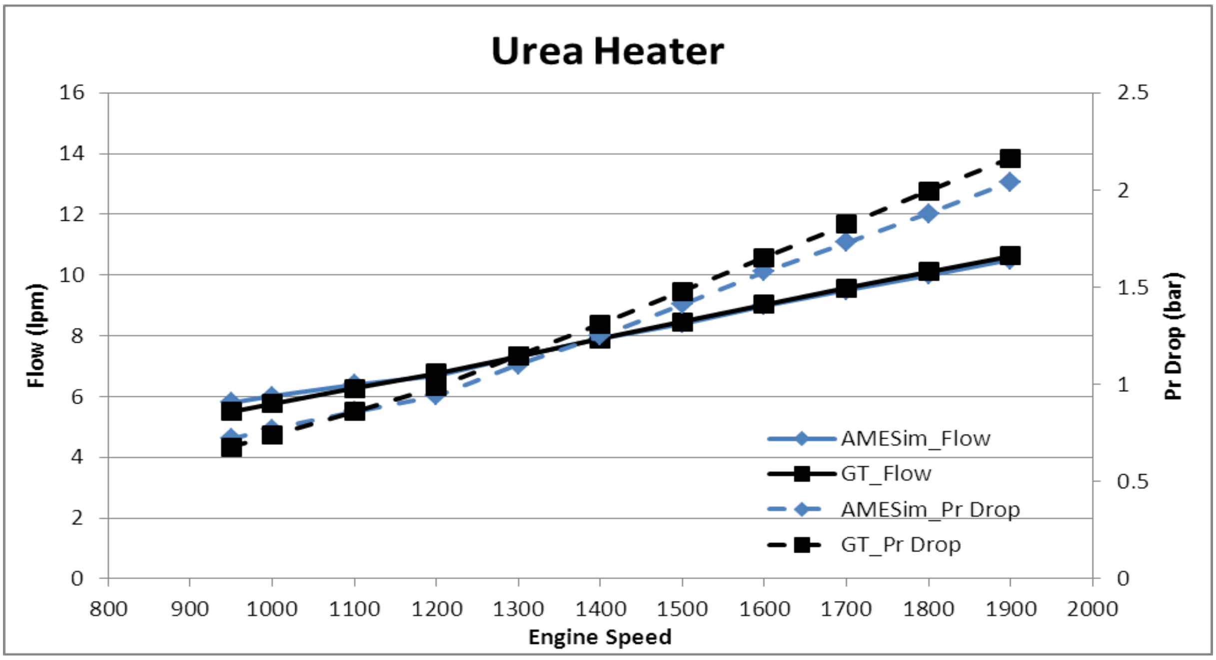


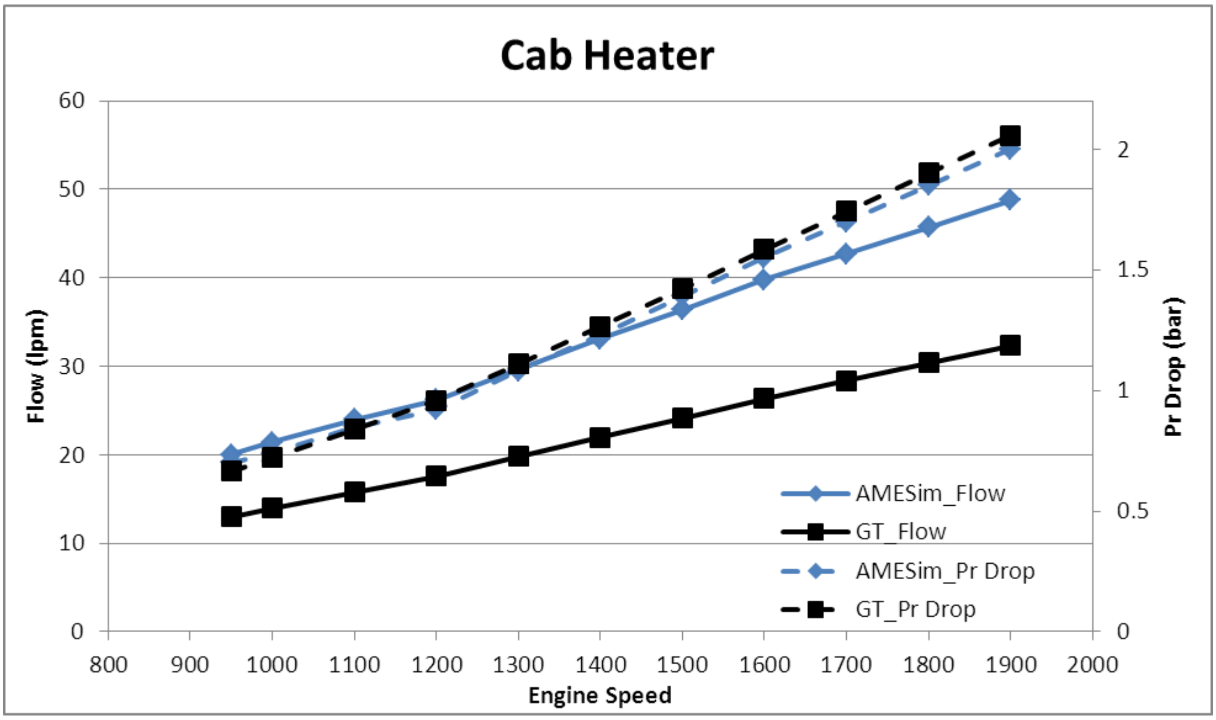
The reason for such a huge divergence in the above air compressor result is the difference in pressure drop values between the two tools. Added to this is the overall lower flow rate caused due to the low flow rate from the coolant pump as shown in GT Cool. The same reason can be attributed to the results of transmission oil cooler shown below.



14.9 Urea Heater and Cab Heater

In urea heater, the difference in flow between the two tools cannot be seen as this component experiences overall low flows. From the pressure drop curve, it can be seen that at lower speeds, there isn't any difference but as the speed increases there is small divergence in the pressure drop value taking place which is attributed due the reasons listed out in the previous sub sections regarding pressure drop in pipes/hoses.





The flow through cab heater shown by GT Cool is low compared to AMESim which is attributed to overall low flow coming from the coolant pump. The reason why the pressure drop curve is matching is because of the additional pressure drop calculated by GT Cool in pipe/hoses connecting the cab heater which inturn actually lifts the GT Cool curve and matches it with AMESim.

14.10 Inlet Manifold Temperature Difference (IMTD)

The IMTD represents the inlet manifold temperature difference, which is the difference in temperature between ambient air and CAC air and EGR mixture in the inlet manifold. IMTD is one way of representation of how much fuel can be saved. The theory behind this is the following. Decrease in air temperature results in an increase in air density. And now since the air density is more, for the same volume of space, one can push in a higher mass of air. In other words, there is more oxygen in the cylinder for combustion to get a higher power output. In simple terms the overall volumetric efficiency of the engine increases. And also it will lower the combustion temperature leading to lower levels of NOx.

The curve shown in Figure 22 gives the IMTD values for each speed as calculate by AMESim and GT Cool. It is calculated as follows

$$IMTD = \frac{(mass\ of\ CAC * Temp\ of\ CAC) + (mass\ of\ EGR * Temp\ of\ EGR)}{mass\ of\ CAC + mass\ of\ EGR} - Ambient\ temperature$$

And hence the difference in IMTD values shown by two modelling tools is because of the reasons stated in the section 14.5. Here also there is divergence from below 1200 rpm and is attributed due to extrapolation limitation in GT Cool.

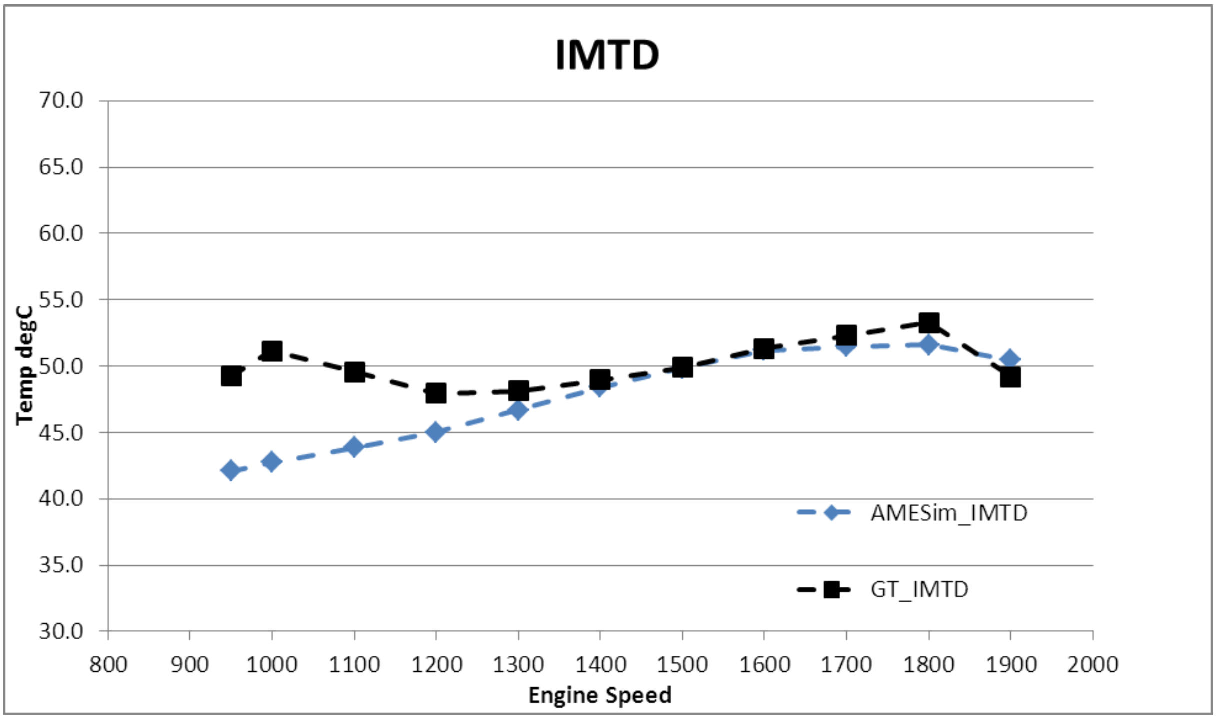


Figure 22: IMTD curve of HDE 16 Euro 6

15 Transient Simulations

As was stated in the previous sections, the transient simulations have been performed in a particular driving cycle which is internally called the Borås-Landvetter cycle. All the factors affecting the convective heat transfer correlations have been incorporated in the model with respect to entrance factor, viscosity factor, surface roughness factor etc. and the convective heat transfer coefficient has been computed.

With respect to GT Cool simulation, the heat transfer correlations were implemented separately and the resulting heat transfer coefficient from the composite correlations mentioned in section 9 were applied on to the pipes as a multiplier.

The heat up time of the engine cylinder wall has also been computed. But because of the limitations with respect to heat load input data mentioned in section 14.2 this does not represent the actual way in which the walls of the 16L DST engine will behave. Never the less, once the input data are collected over the whole engine operating range, this model can be used well to understand and predict the warm-up behaviour.

It is pointed out that some graphs (graphs with X- axis representing time in seconds) shown in the transient simulation results section should not be read in terms of absolute numbers, but in terms of how the warm-up behaviour and thermostat lift fluctuations occur for a transient cycle. It is also pointed out that, once the correct input combustion data is available and when the model is run on those data, the results can vary significantly from the results shown in this report for this particular engine HDEP 16 DST.

As was seen in steady state results, there is a difference of coolant flow rate from coolant pump between both modelling tools because of the way in which both the tools calculate the pressure difference between inlet and outlet of the coolant pump, the same kind of results will be seen for the transient simulations. Hence, only few critical components have been shown with respect to flow and pressure drop components. The most important and critical results from the transient simulations have been discussed in the subsequent sections.

One of the important differences between AMESim and GT Cool was that, AMESim stores and prints data right from the initial input, in this case (transient simulation) 0 seconds. But GT Cool stores and prints result based on the time step of the simulation. Here it is 0.1 seconds. Hence all the results shown in the next section have a shift of 0.1 seconds but for comparison purpose, this shift has been ignored and the results are plotted from 0 to 3680 seconds. But because of this, there is minor and in some points major difference between the results.

16 Transient Simulations on Borås-Landvetter cycle - Results

16.1 Coolant pump

The flow rate and pressure rise across the coolant pump is shown in Figure 23. It is pointed out that the reasons stated for the difference in the steady state results shown by the two modelling tools holds good and valid in this case too.

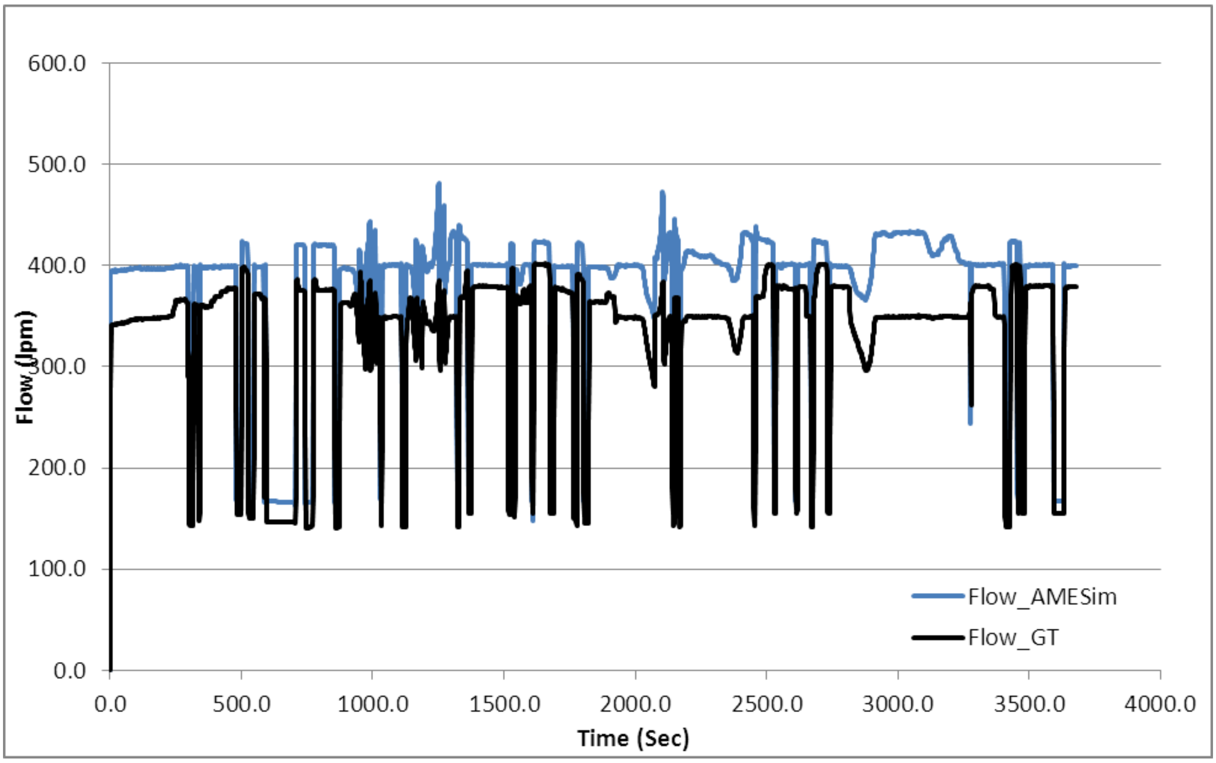
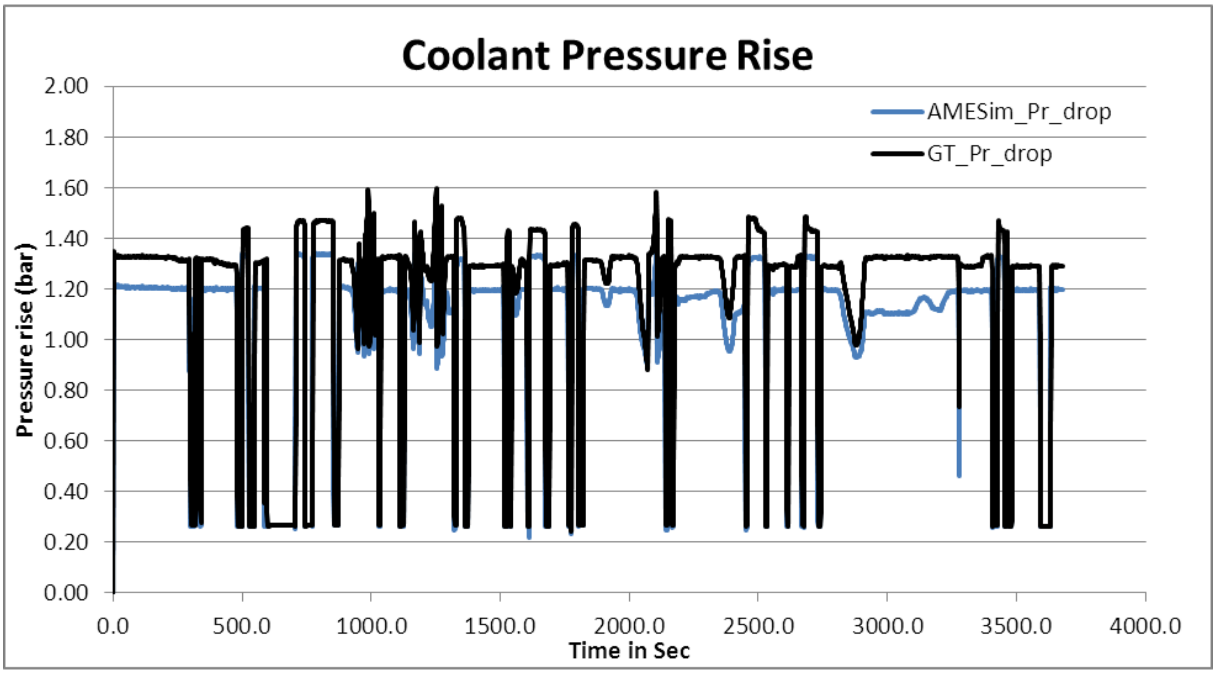


Figure 23: Coolant pump flow - Transient cycle

The model is run for duration of 3680 seconds which covers the time between Borås and Landvetter. Apart from the flow difference which is there because of excess pressure drop calculated by GT Cool as mentioned in the stady state results section, the curves follow each other closely except at few points.

GT Cool, starts the model from initial time setting but while storing the results, it stores the data calculated for the first time step based on the intial condition. But AMESim on the other hand calculates and stores the data for the intial starting condition as well. This is not a big difference between the modelling tool, but it's the way of implementation of the modelling tool's code and preferences.

But this difference is basically there in all the elements of the cooling system and contributes to the difference between the two tools apart from the flow rate differnce as mentioned earlier.



16.2 Thermostat

The important results one can analyse from the transient simulations is that of the thermostat valve lift and flow. The transient cycle was run with a slightly open thermostat in AMESim tool because of difficulty in the convergence. In reality there will always be some amount of leakage through thermostat because of absence of perfect sealing. But the GT Cool model was run without any initial opening as the model worked well without any problem.

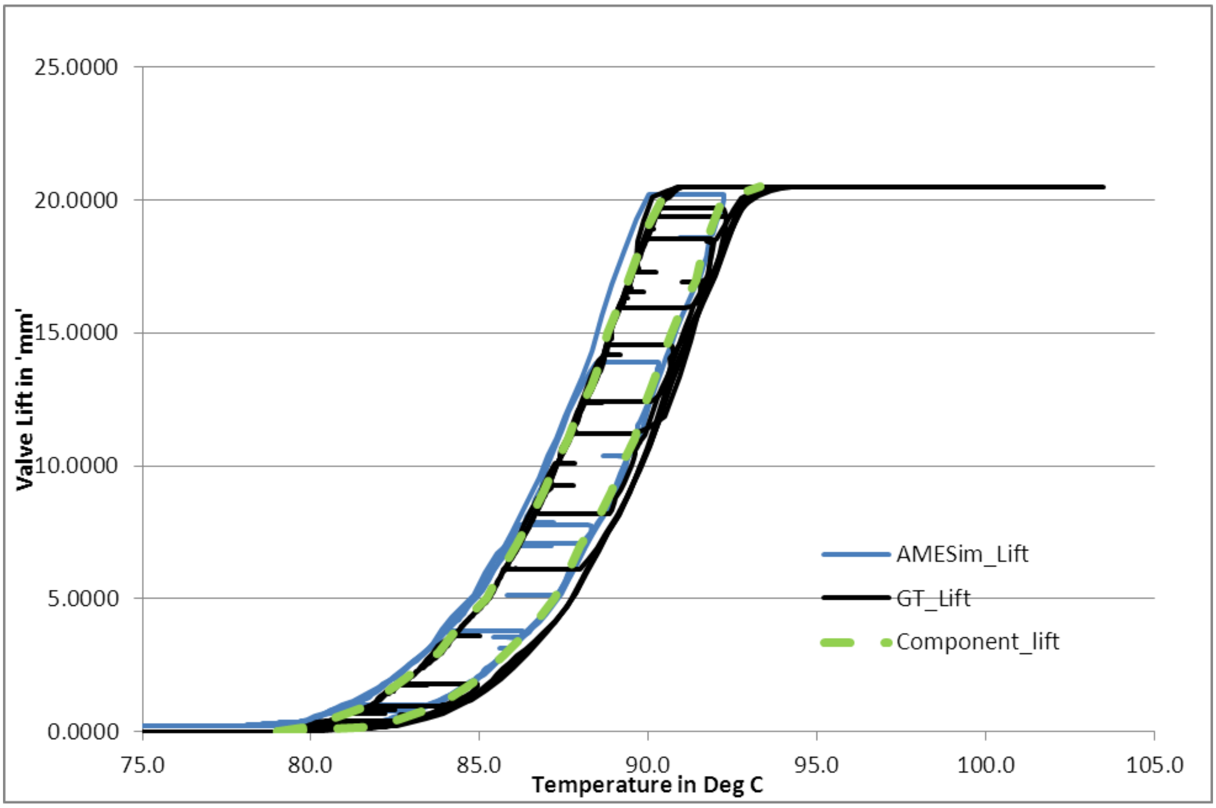


Figure 24: Thermostat lift fluctuation and hysteresis curve

The curve shown in Figure 24 makes a comparison of valve lift with respect to coolant temperature between AMESim, GT Cool and the actual component as tested by the supplier of thermostat. The component curve sets the bounds for the valve lift Vs temperature. If we look at the result closely, in the result shown by GT Cool, the thermostat valve lifts up and down too often when compared with AMESim results. The reason for this is the varying coolant inlet temperature in thermostat. The curve showed in Figure 25 shows the temperature as calculated by the two tools.

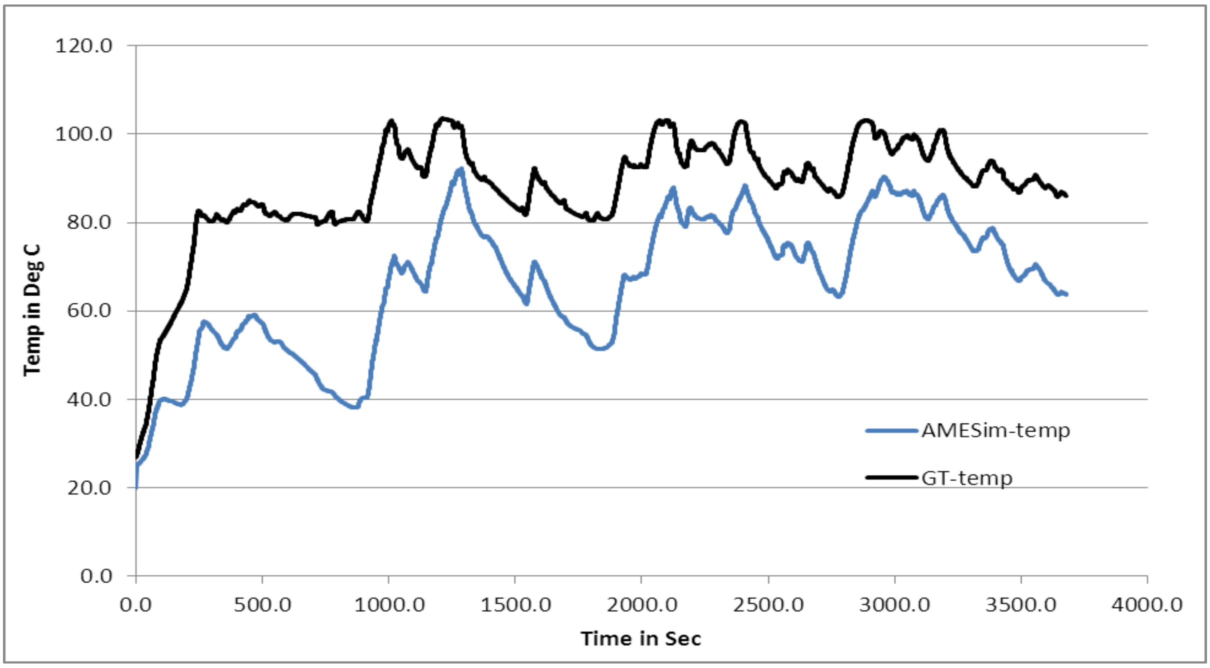


Figure 25: Thermostat coolant inlet temperature

The coolant temperature and wall temperature are closely related and this is discussed in detail in section 16.5. The results shown by AMESim and GT Cool are very smooth and the phenomenon of hysteresis is clearly depicted as shown in Figure 26 below.

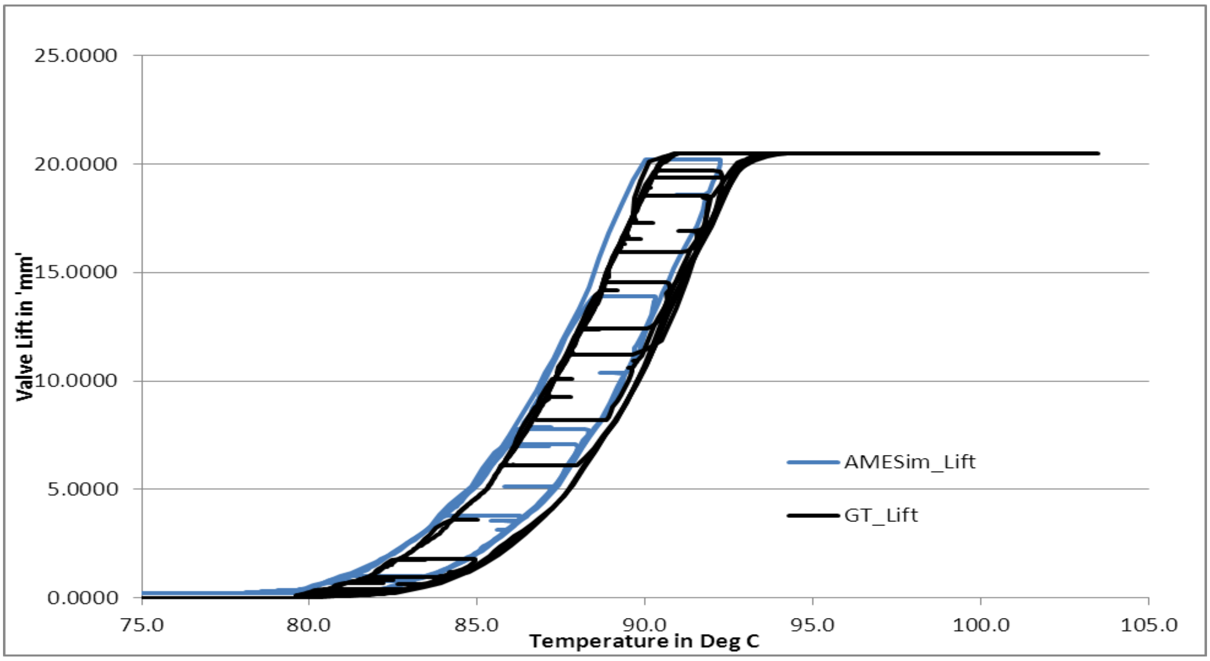


Figure 26: Thermostat lift fluctuation - AMESim Vs GT Cool

Hysteresis is a phenomenon which characterises a wax type thermostat. It's the hysteresis in a thermostat which results in a different valve opening and closing temperatures. Ideally, the thermostat should follow a single curve (temp vs valve lift) for both opening and closing, but in reality there is always a lag in this, and for opening the thermostat follows a particular curve and for closing it follows another curve as is shown in section 11 while describing the thermostat component.

A critical observation can be made from the thermostat lift curve as shown above. The curve depicts the difference between the two tools. It can be seen that there is a shift in opening and closing temperatures. The difference is about 0.5 to 1 degree Celsius. This difference can be resulting from the way the thermostat element reads the temperature and in turn interprets the lift vs temperature data.

The curve shown in Figure 27 below shows the flow across the thermostat towards the radiator.

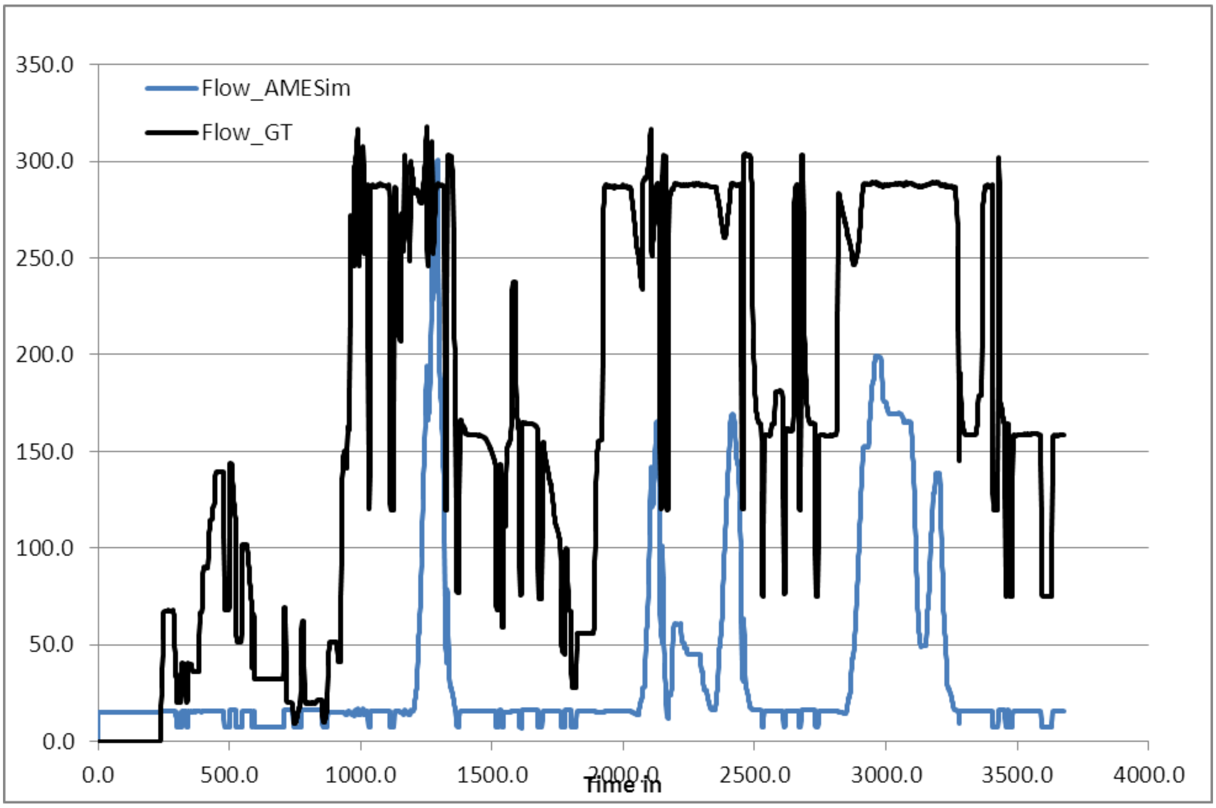
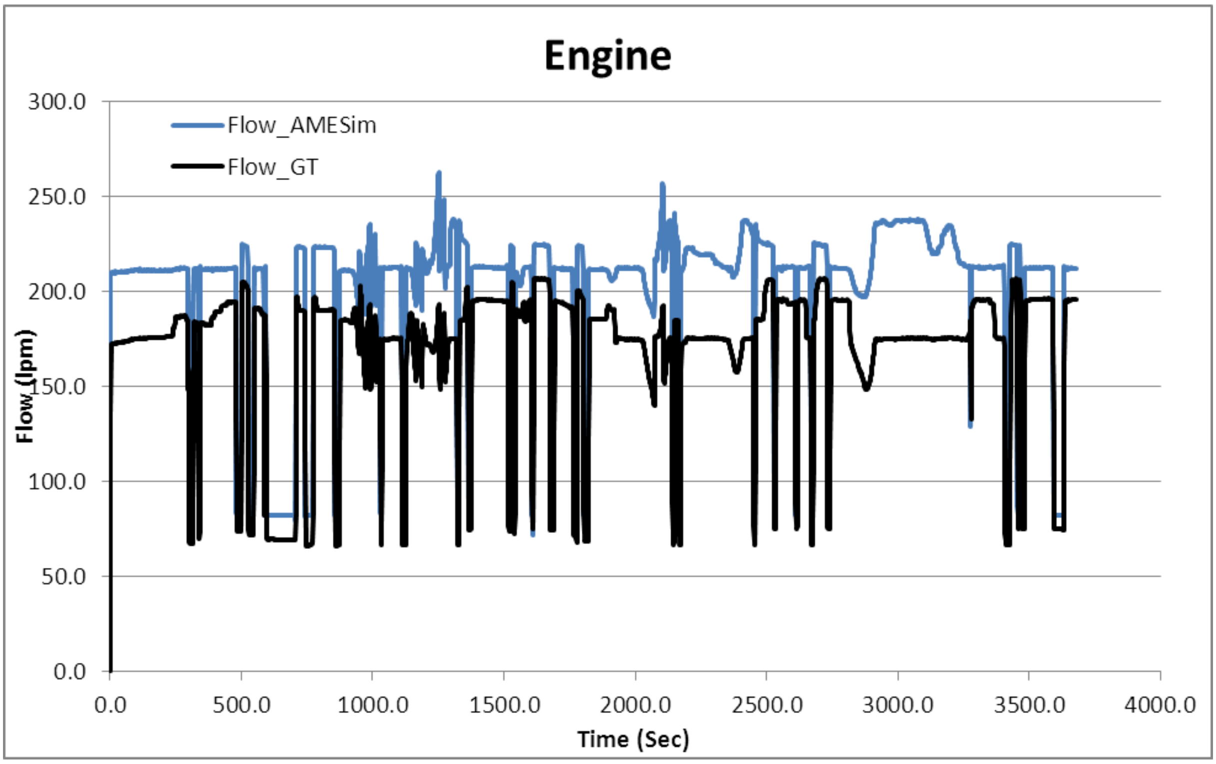


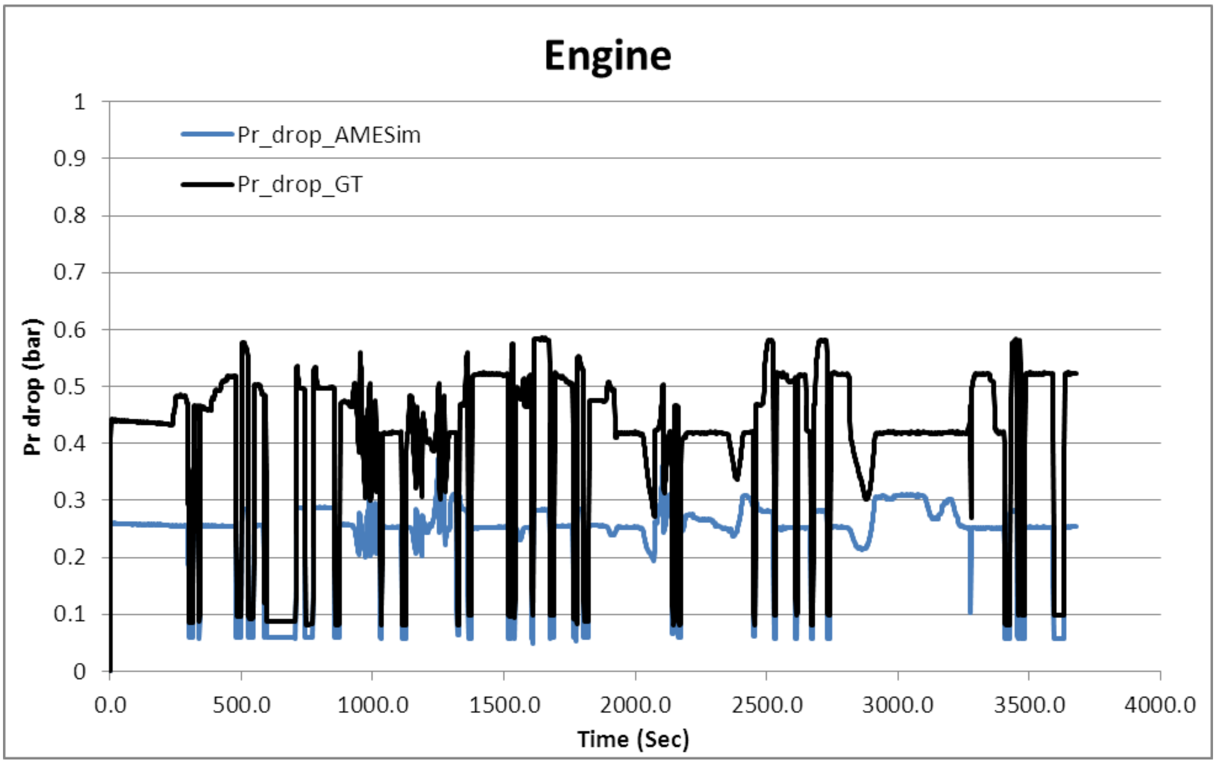
Figure 27: Coolant flow through Radiator

The difference in flow is directly controlled by the valve lift of the thermostat which in turn is controlled by the coolant temperature. It can be seen that, the flow rate shown by GT Cool undergoes lot of fluctuation, in other words, the coolant temperature reaches the thermostat valve opening temperature quite often during the whole cycle as compared to that shown by AMESim. This is because of the heat load variation between the two tools as is discussed in section 16.6.

16.3 Engine

The coolant flow rate and pressure drop across the engine block and cylinder is shown below. The curves shown below more or less follows the same coolant pump curve pattern.





16.4 Engine Speed and Torque

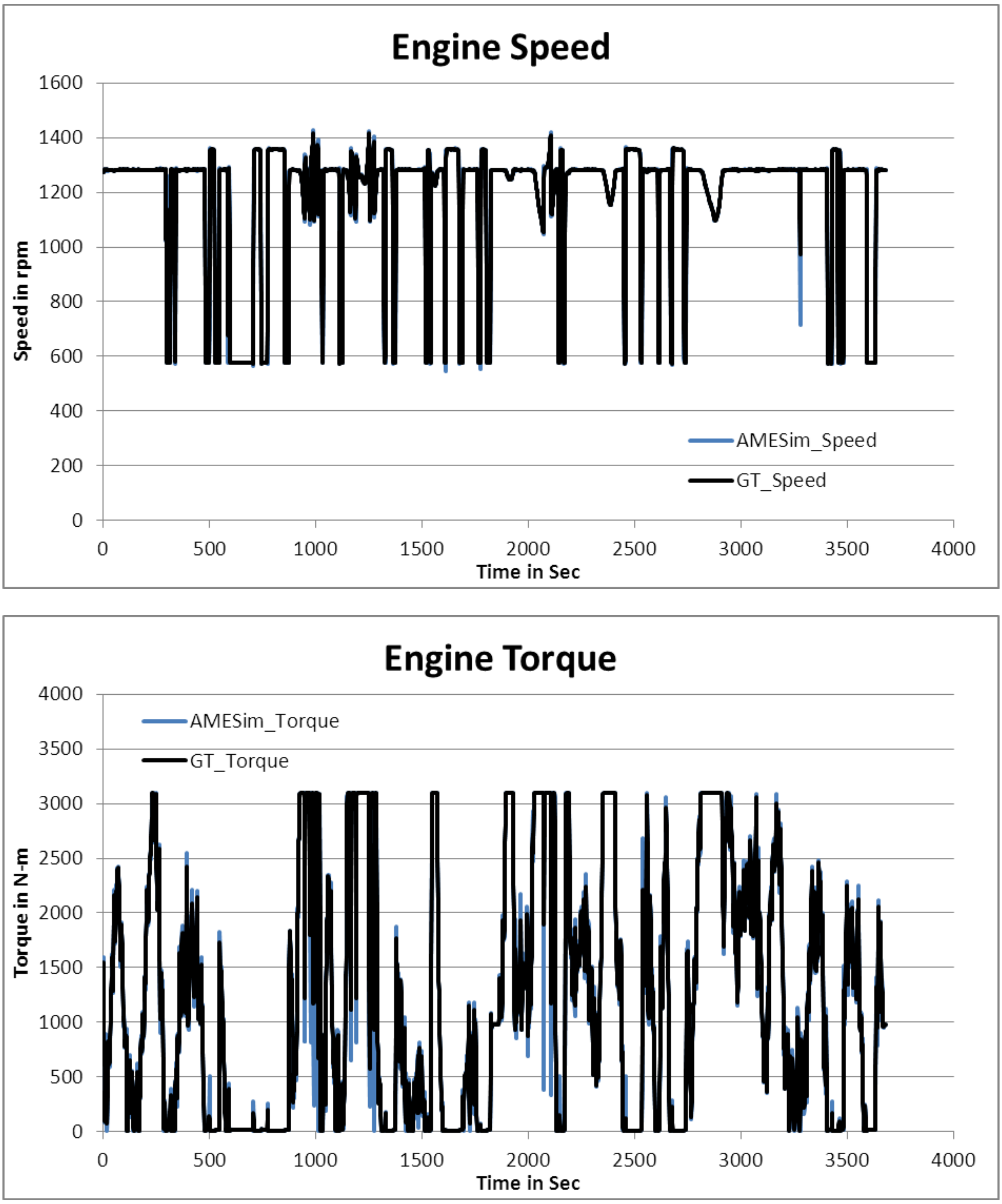


Figure 28: Engine torque- Transient cycle

The way GT Cool stores data can be easily understood from the curve shown in Figure 28. There is not a big difference in terms of reading speed as can be seen from the above graph considering the data storage procedure employed by GT Cool, but there appears to be a major difference in Torque values at some points in the drive cycle.

16.5 Engine cylinder head and Liner warm-up

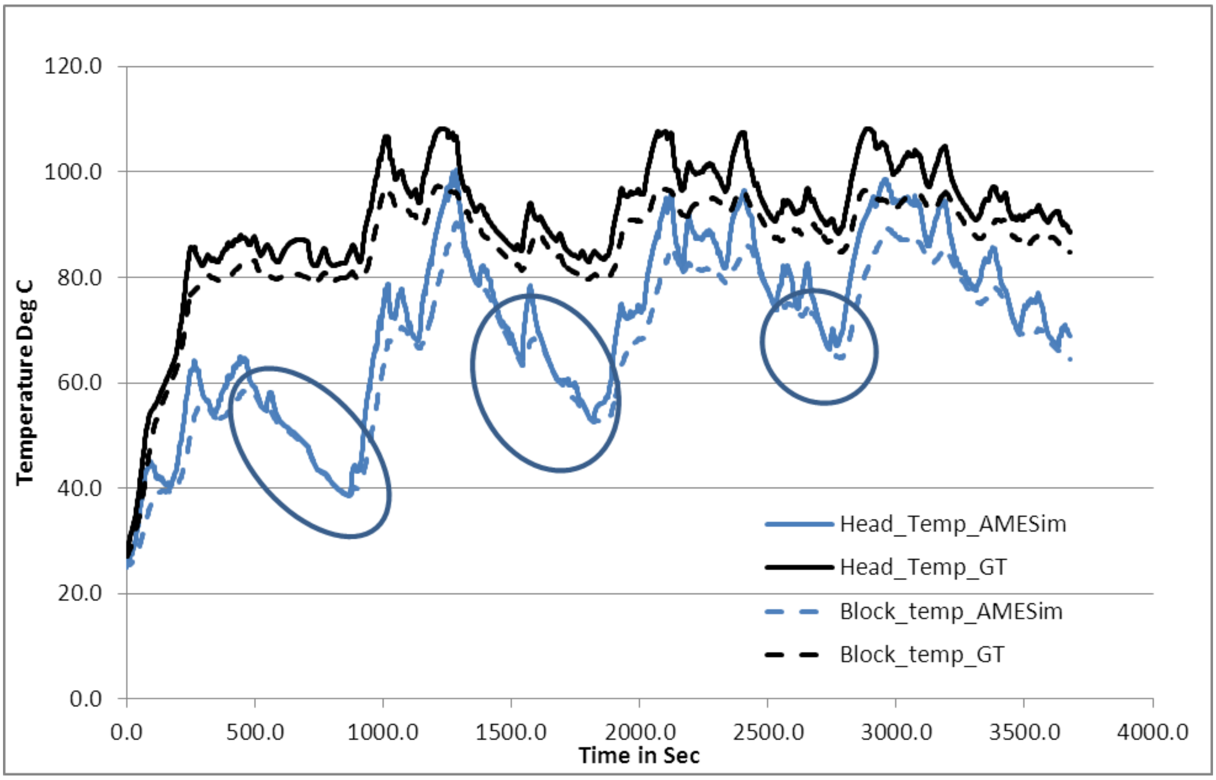


Figure 29: Engine Warm - Temperature profile

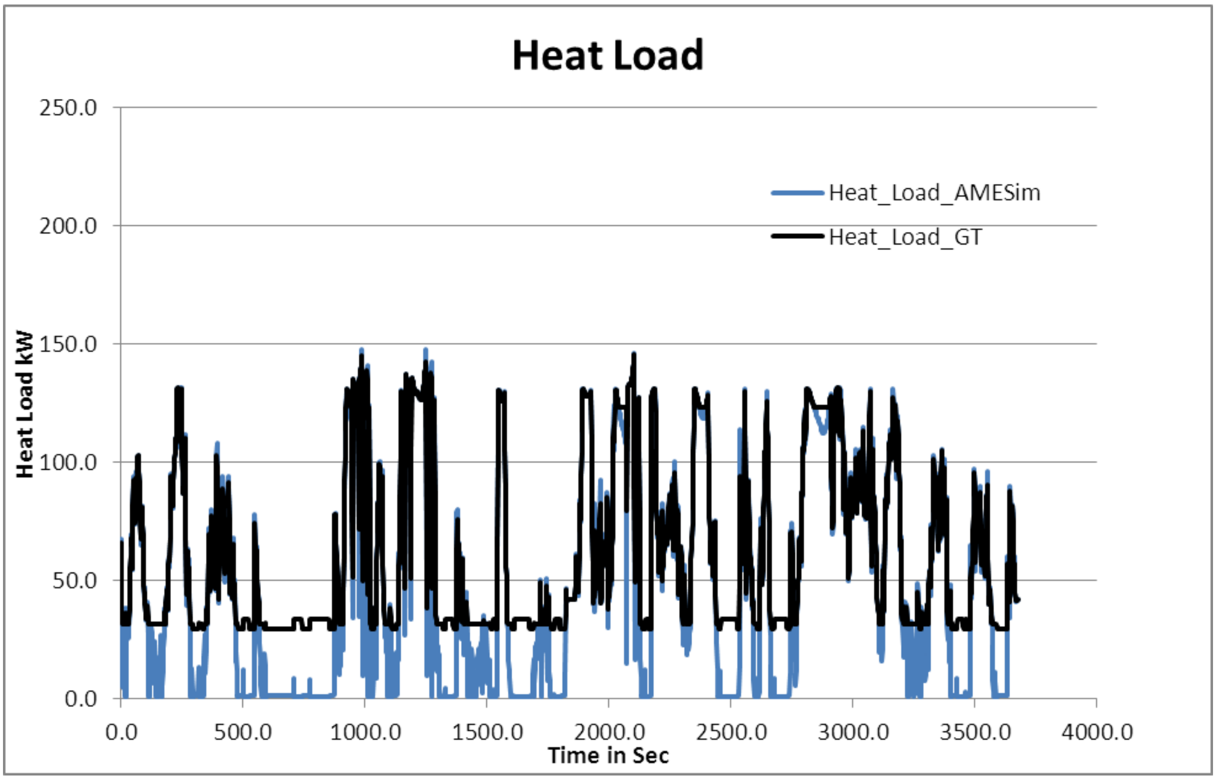
The graph in Figure 29 shows the warm-up behaviour of the engine cylinder head and block (Cylinder Liner). There is a very peculiar phenomenon which can be seen from the results shown by AMESim as is marked by circles in the graph. It can be seen that when the heat load decreases, the temperature also decreases. But if we look at the temperature one can see that downward trend corresponds to the closed thermostat, as the thermostat starts to open from around 79 degree Celsius. In other words, during these times the thermostat is closed and all the coolant should flow through the by-pass port and no coolant should flow through the radiator and hence should maintain the temperature.

In other words, the system should behave as an adiabatic system and hence even when the heat load decreases, the wall should retain the temperature from the previous cycle. The reason for this is the thermostat setting in AMESim tool. If we look at the thermostat flow result in the section 16.2, Figure 27 we can see that the thermostat is kept slightly open. This was done intentionally because of the convergence problem faced during AMESim simulation.

But the results from GT Cool, predict this phenomenon perfectly and when the thermostat is closed, the system behaves as an adiabatic system, and wall temperature is retained from previous cycles.

By analysing this warn-up trend the coolant flow rate can be adjusted during the initial time period of this cycle where the wall temperature doesn't come to the operating temperature of the engine. The engine wall temperature should reach operating temperature as soon as possible because it is during this the engine operates in it lowest efficient point because of increased friction and inefficient combustion because of suboptimal temperature surrounding the combustion chamber.

16.6 Heat Load



As was discussed in the steady state simulations, GT Cool does not extrapolate the heat load values outside the boundary data set in the map and instead outputs the boundary data set in the map for all inputs outside the boundary data. In this case, the boundary is set in terms of torque and engine speed. It can be seen that for all values within this bounds, both the tools gives similar results, but for inputs outside the bounds, both the tools follows different procedure.

From the graph it can be seen that during the idle conditions, AMESim extrapolates the input value outside the boundary data and hence the heat load goes around 10 kW, but GT Cool, because of the extrapolation process, keeps the output heat load close to 35 kW.

This difference basically results in GT Cool showing higher wall temperature and coolant temperature as discussed in previous sub sections and because of this higher temperature the thermostat valve lifts quite often during the entire cycle.

16.7 Convective Heat Exchange Coefficient

The Convective heat transfer coefficient calculated using the composite equation which includes heat transfer enhancement factors like, viscosity factor, surface roughness factor etc. which have been discussed in detail in section 9. The heat transfer coefficient calculated for engine cylinder head and block separately are shown in Figure 30 and 31 below.

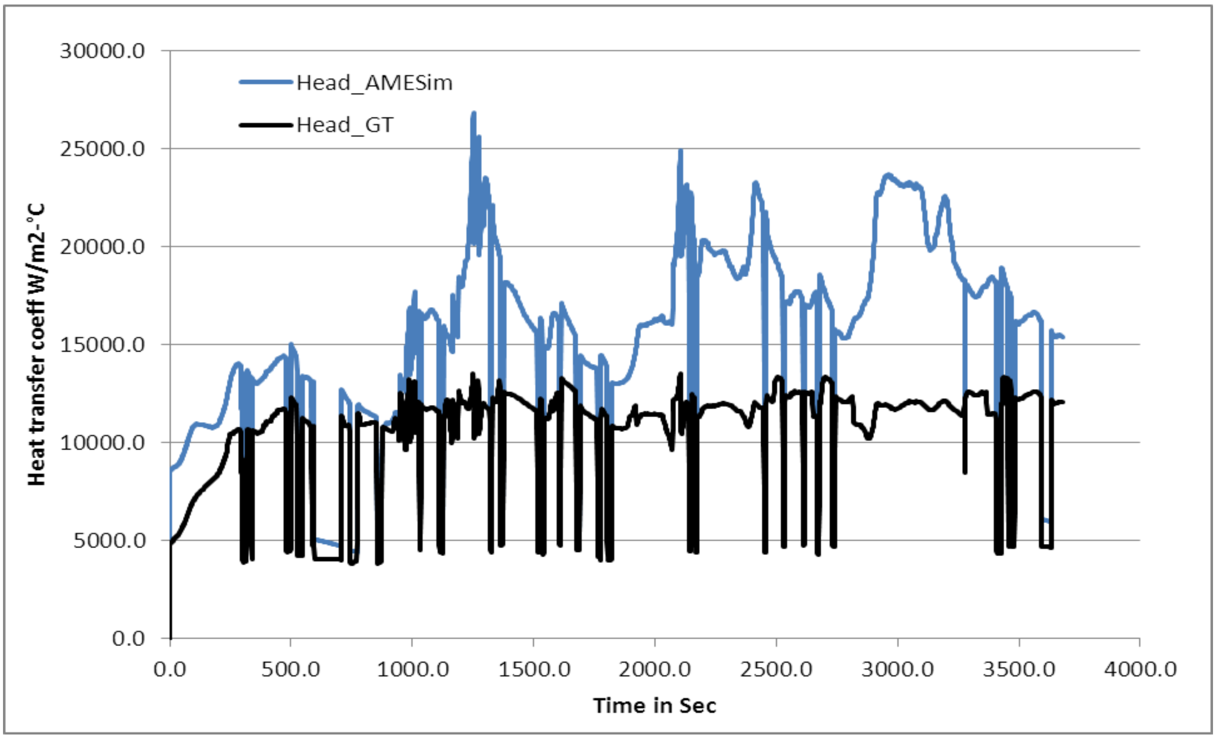


Figure 30: Heat Transfer Coeff - Cyliner Head

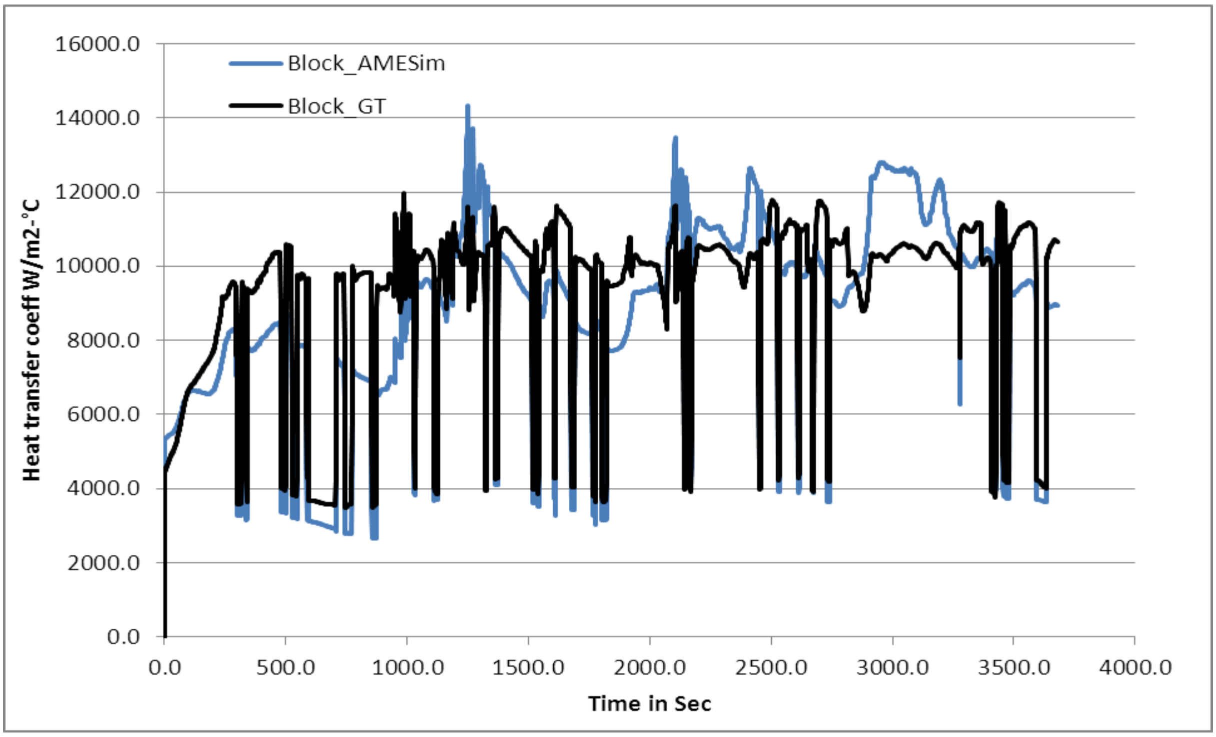


Figure 31: Heat Transfer Coeff - Cyliner Block (Liner)

The heat transfer coefficient basically depends on the Reynolds number which in turn depends on the velocity of the coolant flow which is related to mass flow rate. Since, AMESim shows a higher mass flow rate, the heat transfer coefficient shown is higher as compared to the result shown by GT Cool when it comes to analysing the results of the engine cylinder head.

However, when we analyse, the engine block, there is a deviation. One can see that, at few places, the results shown by GT Cool are higher and in few places, the results shown by AMESim are greater. This is related to mass flow rate through the engine block at particular instances of the drive cycle as calculated by AMESim and GTCool.

The graph in Figure 32 shows the difference in heat transfer coefficient predictions for the cylinder head between the standard Dittus-Boelter correlation and the composite correlation as calculated by AMESim.

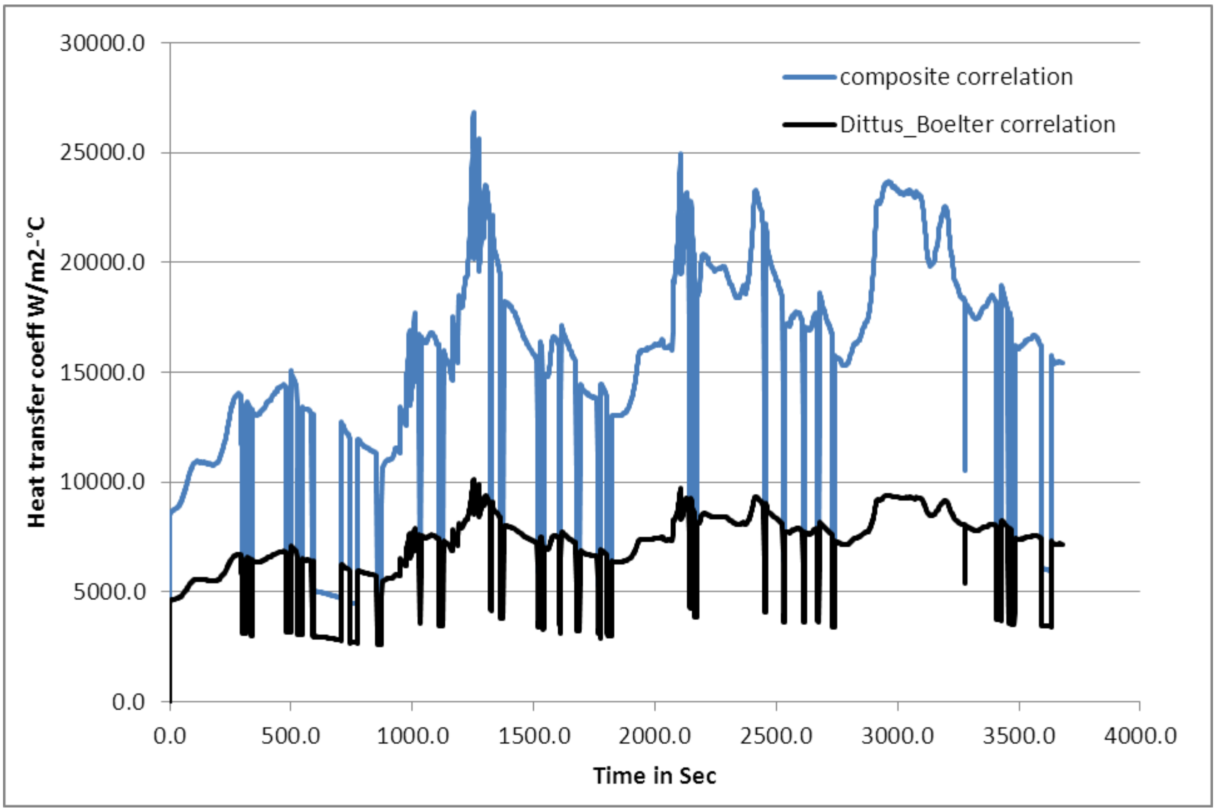


Figure 32: Standard Heat transfer coeff Vs Composite correlation – For Cylinder Head

The difference between standard correlation and composite correlation is almost around 50% and hence has a significant impact on the overall heat transfer phenomenon in the cooling system.

17 Conclusions

The cooling system model for both steady and transient cycle has been built for HDEP 16L DST engine. The model has been built in two different 1-D CFD tools namely AMESim and GT Cool. The results from both the models have been compared and certain conclusions are drawn below. Because of the unavailability of the engine, the results from the model could not be compared with physical tests.

- By analysing the results, one critical conclusion can be drawn with respect to GT Cool tool. The main difference between the results shown by the two modelling tools is the coolant flow rate from the pump. By investigating, it was seen that GT Cool shows an increased pressure drop in pipes/hoses when compared with AMESim. This basically stems from the definition of Reynolds number for calculating friction in pipes/hoses in both the tools. Based on Reynolds number, a flow regime can be either laminar or turbulent and for a laminar region the pressure drop is directly proportional to flow velocity. On the other hand, for turbulent region it is proportional to the square of the flow velocity.

  This results in lowered coolant flow rate from the pump as shown by GT Cool. It will be very difficult to point out which tool is predicting the correct behaviour and hence the model needs to be correlated with physical tests on the engine.

- Results from transient simulation were very useful particularly in analysing the thermostat in terms of valve lift with respect to temperature, but because of unavailability of complete set of input data a final conclusion couldn’t be drawn out.

  Similarly, the engine warm up behaviour can also be better analysed and based on this the coolant flow rate through the engine block can be controlled. Although because of the slightly open thermostat setting in AMESim, this warm up trend couldn’t be analysed perfectly, but GT Cool on the other hand predicts warm up behaviour in a good way because of the closed thermostat setting in the model.

- The heat transfer coefficient enhancement factors were considered and the same were implemented in both the modelling tools and the same were compared with default heat transfer coefficient calculated from standard Dittus-Boelter correlation. Investigative studies done by researchers on this reveals that the standard correlation under predicts the heat transfer coefficient by around 50% and the use of the composite correlation as is implemented in this thesis work under predicts the heat transfer coefficient by around 20% [5].

- The modelling of air side is automatically taken care while building the model by placing end environments and radiator elements in the model in GT Cool, but one needs to build a specific model for modelling the air side in AMESim.

- Calibration of the model took a lot of time in AMESim but in GT Cool it was fairly easy comparatively.

- When it comes to the implementation of concepts particularly with respect to control system of the cooling system, AMESim is slightly better than GT Cool in terms of ease of implementation.

All the results shown above are computational results and the same needs to be correlated with physical tests and based on that further fine tuning of the model needs to performed.

18 Future Work

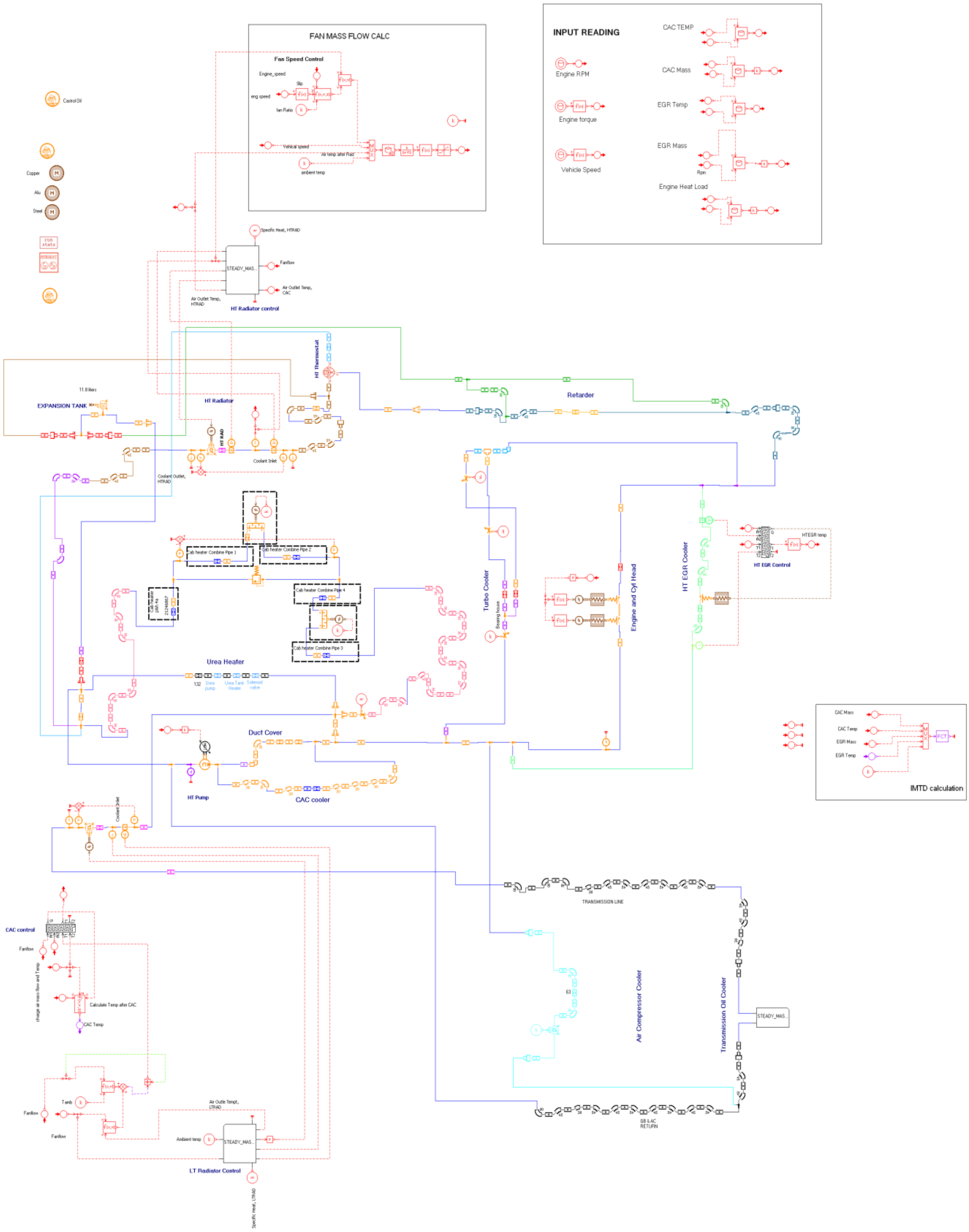
The model developed in both AMESim and GT Cool runs for both Steady and transient simulations. Based on the results discussed in report with respect to warm up behaviour and thermostat valve lift behaviour, the developed model can be better utilised if certain control strategies can be developed to control the pump flow rate based on the requirement of the engine and other accessories of the truck cooling system.

Similarly, the use of electronic thermostat can be investigated and certain control strategies can be implemented based on the coolant temperature and certain other factors. The fan model implemented in the current cooling system model is a very basic model which basically runs on 3 speed regimes based on the coolant temperature.

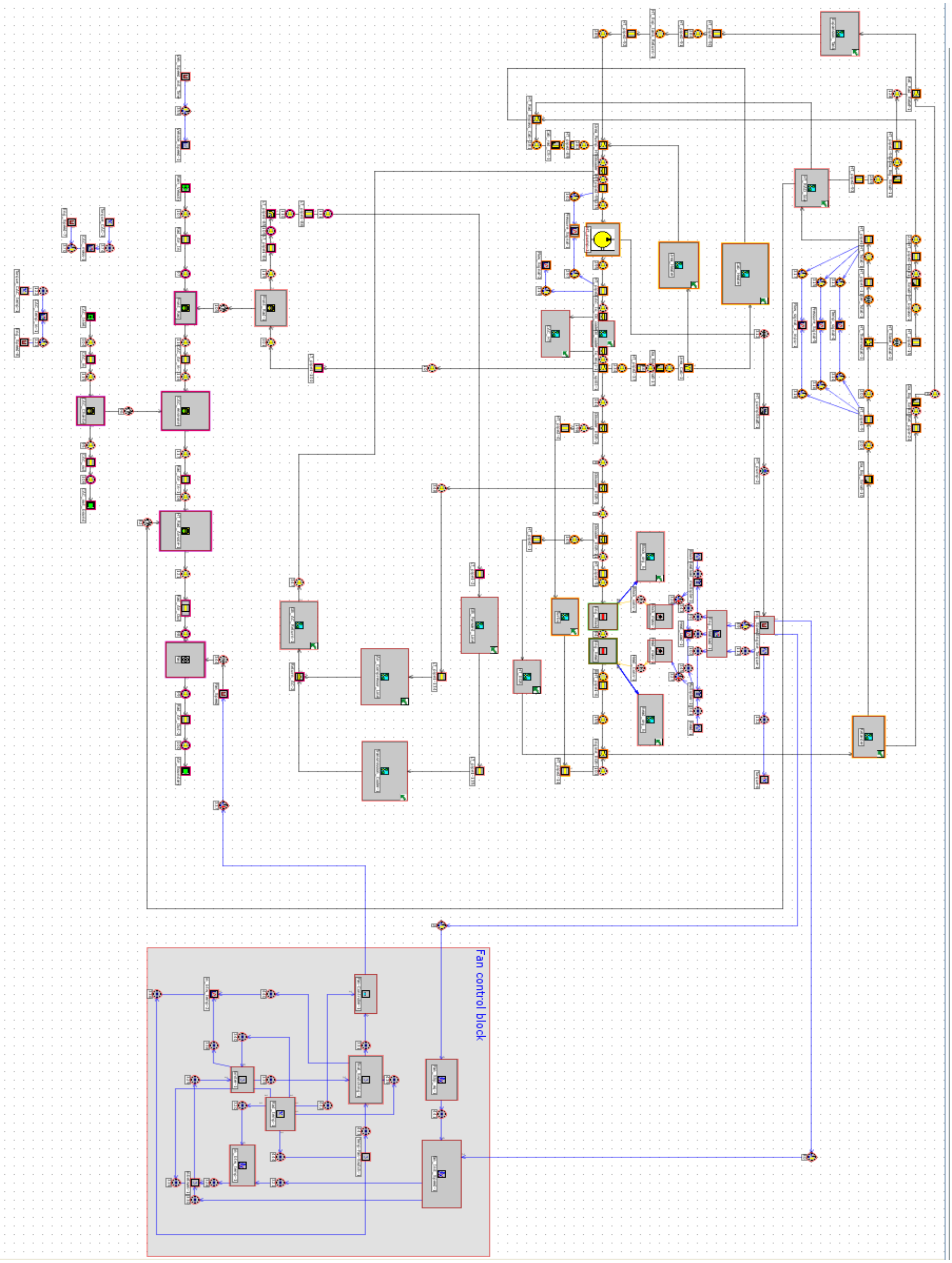
Another important thing to be done is to do physical test on the engine to correlate the results from this model and subsequently tweak the model and fine tune the same to get proper results.

19 Appendix

19.1 Appendix A – AMESim model



19.2 Appendix B - GT Cool Model



20 References

1. Martin W. Wambsganss, ‘Thermal Management concepts for Higher-Efficiency heavy vehicles’, SAE paper No. 1999-01-2240.

2. Rainer Krafft, Andreas Wolf, Wolfgang Faller, ‘Electromagnetic water pump clutch: Working principle, Design strategies and applications for heavy duty vehicles’, SAE paper No. 2007-01-4260

3. Clive Hughes, Chad Mitts, Frederic Jacquelin and Marc Wiseman, ‘Heavy Duty truck cooling system design using co-simulation’, SAE paper No. 2001-01-1707

4. Kyu Hyun Lee and Joo Heon Lee, ‘Development of a continuously variable speed viscous fan clutch for engine cooling system’, SAE paper No. 980838

5. K Robinson, J G Hawley, G P Hammond and N J Owen, ‘Convective coolant heat transfer in internal combustion engines’, Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering February 1, 2003 vol. 217 no. 2 133-146

6. AMESim Rev 10, Tutorial Manual

7. GT Suite V 7.1, Cooling Systems and Thermal management Applications Manual.

8. ‘Development of a Continuously Variable Speed Viscous Fan Clutch for Engine Cooling System’, SAE Paper No. 980838

9. Volvo Database

10. Fundamentals of Thermal-Fluid Sciences by Yunus A. Cengel, John M. Cimbala and Robert H. Turner (2007)