Abstract

***Title**: Transient simulations of a gasoline engine cooling system using Flowmaster2*

The car industry of today is exposed to hard competition. For every new car model the requirements of the engine performance and cooling capacity is increased. To increase the cooling capacity the components in the cooling system needs to be improved and the performance to be evaluated. An often used method to predict the behaviour of components and complete systems during its development is to make physical tests on a concept. This process is very time consuming and costly. An alternative technique to reduce these factors is to use a computer aided simulation tool for simulations of individual components and the total systems. The advantage of this approach is that concepts can easily be evaluated and changed if necessary.

The aim of this work is to improve and analyse a virtual model for transient heat transfer calculations of an engine cooling system in the simulation software Flowmaster. A main part of the work is to implement the air cooling circuit in the simulation model and to study the effects of the air stream through the cooling package in the front of the vehicle. To be able to simulate the air flow, CFD-simulations are consulted and the produced results are used as a boundary condition in the Flowmaster model.

The analysis consists of comparisons between the Flowmaster model and results from physical tests in a wind tunnel. The simulation results in Flowmaster show a relatively good correspondence with the wind tunnel tests. Deviations are analysed and evaluated by performing sensitivity analyses of parameters of interest.

Preface

This master thesis was executed as a final work in the Master of Science education at Chalmers University of Technology in Gothenburg. The work is done for the Volvo Car Corporation Cooling System Department in Gothenburg. It was carried out during the autumn/winter of 2005/2006.

We specially would like to thank our mentor, Peter Norin at Volvo Cars and our examiner Håkan Nilsson (Assistant Professor) at the Department of Applied Mechanics at Chalmers for their support and guidance throughout the thesis.

We would like to express our gratitude to the personnel at the Cooling System Department for sharing their knowledge and experience with us. We would also like to thank Benjamin Olaison, Cecilia Nilsson, Ulrik Albrecht, Jacob Krantz, Kaj Johansson and Rikard Hansson for the assistance and support.

Finally, we would like to thank Jonas Hermansson at Volvo Cars, support engineer Johan Henriksson at Flowmaster and John Moffat at Ford Motor Company for the help and support during the development of the Flowmaster model.

Nomenclature

Acronyms

CAC=Charge Air Cooler
CFD=Computational Fluid Dynamic
EOC=Engine Oil Cooler
NA/T= Naturally Aspirated/Turbo charged
SI6= Six cylinder gasoline engine
SI6 NA= Naturally Aspirated gasoline engine
SI6 T=Turbocharged gasoline engine
WTOC=Water Transmission Oil Cooler

Nomenclature

A=Cross-section area of the component $[\mathrm{m}^2]$
c_p=Specific heat capacity at constant pressure [$J/kg \cdot K$]
f =Darcy friction factor [-]
H=Head [m]
K=Loss coefficient [-]
K_r=Ram coefficient [-]
$\dot{m}$=Mass flow rate [kg/s]
M=Pump torque [Nm]
P=Pressure [Pa]
ΔP =Total pressure increase across pump [Pa]
ΔP_{Ram}=Ram pressure [Pa]
Q=Volumetric flow rate $[\mathrm{m}^3/\mathrm{s}]$
q'' =The convective heat flux $[\mathrm{W/m}^2]$
q_r=Radiator airflow based dynamic pressure [$kg/m \cdot s$]
q_0=Free stream dynamic pressure [$kg/m \cdot s$]
Re=Reynolds number [-]
T=Temperature [K]
T_s=Surface temperature [K]
T_∞=Fluid temperature [K]
T_0=Reference temperature of fluid (if set) or Ambient temperature of the network [K]
$(\Delta p_i)_{CS}$=Internal interference between the cooling-package components [Pa]
V_v=Vehicle velocity [m/s]
V_r=Radiator air velocity [m/s]
ρ=Fluid density $[\mathrm{kg/m}^3]$
η=Efficiency [-]
ω=Rotational speed [rev/s]

Subscripts

bay=Engine bay or underhood compartment
C=Cold stream
CS=Cooling subsystem
con=Condenser
(*fan+sh)*=Fan and shroud
H=Hot stream

i=Inlet
inlet=Front-end cooling opening
o=Outlet
rad=Radiator
ram=Free stream energy recovery by the inlet when the vehicle is moving
t=Total
ub=Underbody

Table of contents

1 Introduction

1.1 Background

One of the challenges the modern car industry is facing is the requirements of shorter development time and higher performance for new car models. An often used method to predict the behaviour of a car during its development is to make physical tests on a concept. This process is very time consuming and costly. An alternative technique to reduce these factors is to use a computer aided simulation tool for simulations of individual components and the total system. The advantage of this approach is that concepts can be easily evaluated and changed if necessary.

In a modern combustion engine only one third of the internal energy of the fuel is converted into useful work. The remaining two thirds of the energy are transformed into waste heat which has to be dispelled in order not to overheat the engine parts. The waste heat is diverted from the engine by the exhaust, the integrated cooling systems and convection and radiation of the engine solid parts. Ultimately the heat is transferred to the surrounding air by various heat transfer processes.

When predicting temperatures and flow rates in an engine cooling circuit, a large number of 3-dimensional CFD-analyses (3D flow simulations) are typically required in order to determine the air flow patterns through the cooling package over a range of vehicle operating conditions. This method is very time consuming due to the construction of 3D models and the large number of CFD-simulations required. An alternative approach is to use a 1-dimensional simulation tool which is much less time consuming and have the capacity to simulate larger components systems. However, this technique requires a characterization of the varying air flow.

1.2 Objectives

The objective of this master thesis has been to build and analyse a virtual model for transient heat transfer calculations of a vehicle cooling system in the 1D fluid flow simulation software *Flowmaster*. A starting point for the work has been models used for fluid flow and steady state heat transfer calculations solely. Our ambition has been to extend these models to be able to simulate heat transfer between components in transient driving cases such as warming up from cold start and different driving cycles.

The goal for the thesis included an implementation of a model simulating the airflow through the bumper and grille in the front of the car and a model of the fan. The Flowmaster model should be verified against physical tests.

1.3 Method

First, the actual cooling system was analysed and the existing Flowmaster models were studied. Components in the Flowmaster models were modified to cope with transient simulations and heat transfer.

To be able to build a model simulating the variation in airflow through the grille with different vehicle velocity, results from CFD-calculations were consulted. These 3D results were coupled to a 1D model by assistance of the *Ram curve* concept [4].

Component characteristics for the charge air cooler, radiator, condenser and fan were studied and implemented in Flowmaster.

Comparing the new Flowmaster model with physical tests requires some engine- and vehicle-specific input data to be matched. Theoretically, if our model was a perfect emulation of the physical vehicle, the utilization of the characteristic parameters would generate the same results as the physical model. The engine/vehicle characteristic parameters are rpm, load (torque) and vehicle speed, and were obtained from physical tests in a wind tunnel where numerous parameters were measured.

When the characteristic data was obtained and applied to the Flowmaster model, the results were analyzed and compared with the physical model. Some of the results of interest are coolant temperature at various nodes, coolant flow oil etc. The divergence between the results was analyzed and evaluated in order to optimize the computer model.

1.4 Limitations

In this project we have focused on the air cooling and its interaction with the coolant cycle and oil cooling cycle.

A model of the coolant circuit already exists in Flowmaster due to previous work.
The oil cooling systems including the engine oil system and the transmission oil system, described in chapter 5, are considered as simplified circuits with outgoing heat fluxes to the coolant circuit.

One part of the project goal was to simulate the transient heating of the solid parts in the engine and cooling fluids, derived from the internal energy in the fuel. This task was not managed due to the limitation of our time frame to a maximum of 20 weeks and lack of necessary physical engine tests.

2 Heat Transfer Theory

Heat transfer is thermal energy in transit due to a temperature difference. Whenever a temperature difference exists in a media or between media, heat transfer must occur. There are three different types of heat transfer processes. When a temperature gradient exists in a stationary medium the term conduction is used to refer to the heat transfer that will occur across the medium. The heat transfer that will occur between a surface and a moving fluid when they are at different temperatures is called convection. The third process of heat transfer is termed thermal radiation. All surfaces of finite temperature emit energy in the form of electromagnetic waves.

In this section the basic heat transfer theory behind the equations in this study will be presented [1], [3].

2.1 Conduction

Conduction may be viewed as the transfer of energy from the more energetic to the less energetic particles of a substance due to interactions between the particles. Energy is related to random translational motions, as well as to internal rotations and vibrational motions of the molecules. Higher temperatures are associated with higher molecular energies, and when neighboring molecules collide a transfer of energy from the more energetic to the less energetic molecules must occur, in the direction of decreasing temperature. For heat conduction, the rate equation is known as *Fourier's law*:

$$\mathbf{q}'' = -k\left(\mathbf{i}\frac{\partial T}{\partial x} + \mathbf{j}\frac{\partial T}{\partial y} + \mathbf{k}\frac{\partial T}{\partial z}\right) \text{ [W/m}^2\text{]}$$ **Equation 2.1**

Where k is the thermal conductivity [W/mK] and T is the temperature.

Heat produced in the engine is transferred in the cylinder block and cylinder head by conduction.

2.2 Convection

The convection heat transfer process is comprised of two mechanisms. Conduction and advection, where advection is energy transferred by the bulk, or macroscopic, motion of the fluid. Convection heat transfer occurs between a fluid in motion and a bounding surface when the two are at different temperatures. At the interface between the surface and the fluid, the fluid velocity is zero (the no-slip condition) and the heat is transferred only by conduction.

The rate equation for convection heat transfer is of the form:

$$q'' = h(T_s - T_\infty) \text{ [W/m}^2\text{]}$$ **Equation 2.2**

where q'', the convective heat flux is proportional to the difference between the surface and fluid temperatures, T_s and T_∞, respectively. This expression is known as *Newton's law of cooling*, and the proportionality constant h [W/m^{2}K] is termed the *convection heat transfer coefficient*. The convection heat transfer coefficient depends on conditions in the boundary layer, which are influenced by surface geometry, the nature of fluid motion, and an assortment of fluid thermodynamic and transport properties.

The conducted heat to the engine solids is convected to the moving cooling fluids circulating in the engine. From the cooling fluids the heat is convected to the air by heat transfer in the heat exchangers. To some extend the engine solid parts are cooled by convection by the passing under hood air flow.

2.3 Radiation

Thermal radiation is energy emitted by a solid or a fluid that is at a finite temperature.

This thesis solely treats heat transfer in cooling systems fluids and does not include radiation from the solid parts of the engine. Heat transfer from the cooling fluids to the surrounding by radiation is assumed to be small compared to heat transfer by conduction and convection. Therefore it is neglected in this report.

2.4 Energy conservation

The first law of thermodynamics, equation 2.3 (*the law of conservation of energy*) is a useful, often essential, tool. The amount of thermal and mechanical energy that enters a control volume, plus the amount of thermal energy that is generated within the control volume, minus the amount of thermal and mechanical energy that leaves the control volume must equal the increase in the amount of energy stored in the control volume.

$$E_{in} + E_g - E_{out} \equiv E_{st}$$ **Equation 2.3**

3 Flowmaster Theory

Flowmaster is a 1D fluid flow simulation software package used to analyse fluid flow conditions in simple and complex piping networks. It provides a graphical virtual environment where it is possible to design, refine and test the entire fluid flow system. The ability to simulate these systems can lead to reduced physical testing costs and a faster time to market.

3.1 The fundamentals of Flowmaster

The Flowmaster single phase steady state and transient modules have been specifically designed for modeling the effects of heat transfer in many application areas. The transient modules also allow transient events to be analyzed [2].

Each Flowmaster component represents a mathematical model of an engineering component. Selected components are connected via nodes to form a network, which forms the actual computer model of the flow system. Data tables, curves and surfaces can be used to define the operation and performance of each component. When you run the analysis, pressure, flow rates and temperature are calculated throughout the network. The possible results obtained in each connection are: volumetric flow, velocity, mass flow rate, temperature, liquid density and liquid viscosity.

Most component results contain the quantity *Total Heat Flow*. This is defined as:

$$\text{Total Heat Flow} = \dot{m} c_p (T - T_0) \qquad \textbf{Equation 3.1}$$

The equation represents the heat content of the fluid relative to T_0.

The governing equations used by Flowmaster are the conservation of mass (equation 3.2), energy (equation 2.3) and momentum (equation 3.3, equation 3.4, and equation 3.5). These are used to calculate the flow into and out of the components as a function of the pressure. Flow is then eliminated from the equations using continuity at each node. This leaves a set of equations which can then be simultaneously solved. The resulting pressures are substituted back into the component equations to calculate new estimates of the flows. Any pressure specifying components impose a pressure at their connecting node. This iterative process is repeated until stable values are achieved.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0 \qquad \textbf{Equation 3.2}$$

$$\rho g_x - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) = \rho \frac{du}{dt} \qquad \textbf{Equation 3.3}$$

$$\rho g_y - \frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) = \rho \frac{dv}{dt} \qquad \textbf{Equation 3.4}$$

$$\rho g_z - \frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) = \rho \frac{dw}{dt} \qquad \textbf{Equation 3.5}$$

These equations stipulate that the same amount of energy, mass and momentum that goes into a control volume must leave it. Equations 3.3, 3.4 and 3.5 are the *Navier-Stokes* equations. From the *Navier-Stokes* equations *Bernoulli's* equation (equation 3.6) can be derived. It is a relation between pressure, velocity and elevation along a streamline [7].

$$\frac{V^2}{2g} + z + \frac{p}{\rho g} = C$$ **Equation 3.6**

In the Bernoulli equation, V is the stream velocity, g is the gravitational acceleration, z is the altitude, p is the static pressure, ρ is the density and C is a constant.

When solving a transient heat transfer problem an iterative solution procedure has to be adopted because of the dependence of the specific heat on the solution. This is incorporated into Flowmaster's main iterative solution procedure i.e., the algorithm for $T_2(t+\delta t)$ will use the current specific heat values, which will progressively get more accurate as Flowmaster converges to a solution.

3.2 Components in Flowmaster

Below follows a description of components used in the Flowmaster model [2].

3.2.1 Sources

Sources provide a boundary condition of user-defined flow or pressure. You must always specify pressure somewhere in the network. Sources are also used to specify temperature in the network. Figure 3.1 illustrates two different kinds of sources in Flowmaster.

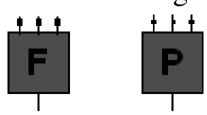

Figure 3.1 Flow source and Pressure source

Flow source

A *flow source* component supplies a constant flow rate either to or from the component, irrespective of the other conditions. The following equation defines the mass flow to the node:

$$\dot{m}_1 = \rho Q$$ **Equation 3.7**

Where $\dot{m}_1$ is the mass flow rate at connection 1.

The nodal pressure is calculated from the pressure term in the energy equations of one or more of the connecting branches.

Pressure source

A *pressure source* applies a constant pressure to the connecting node. The mass flow rate from the *pressure source* to the connecting node is determined by continuity at the node, when all other mass flow rates have been determined by back substitution into energy equation for the corresponding components.

3.2.2 Heat Exchangers

Heat exchangers represent components that model the transfer of heat.

Heater-cooler

This component (Figure 3.2) is used to model a simple loss for which the thermal duty can be specified. It has a hydraulic resistance that is specified by the flow area and a loss coefficient. The pressure/flow equation is defined as follows:

$$P_1 - P_2 = \frac{K\dot{m}_2|\dot{m}_2|}{2A^2\rho}$$ **Equation 3.8**

Where P_1 is the pressure at node 1, P_2 the pressure at node 2 and $\dot{m}_2$ the mass flow rate to node 2.

The loss coefficient K is based on the mean flow velocity, i.e.;

$$\Delta P = K\frac{1}{2}\rho V^2$$ **Equation 3.9**

The thermal analysis is based on the steady flow energy equation, which for constant c_p becomes:

$$Q = c_p \dot{m}(T_2 - T_1) \ \ [\text{kJ}/s]$$ **Equation 3.10**

Where Q is the heat input, T_1 the inlet temperature and T_2 the outlet temperature.

If c_p varies with temperature, then c_p is integrated over the temperature range.

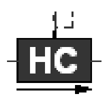

Figure 3.2 Heater-cooler heat exchanger

Thermal

The *thermal* component (Figure 3.3) has four fluid connections and it models heat exchange between two streams in a network. There is no mixing of the two fluids. The pressure loss of both flow parts is specified by defining a loss coefficient and characteristic flow area for each flow i.e. in the same way as for the *heater-cooler* component.

The thermal analysis is based on the steady flow energy equation for each stream with all the energy leaving the hot stream being transferred to the cold stream. Thus for constant specific heat values:

$$(\dot{m}\bar{c}_p)_H(T_{Hi} - T_{Ho}) + \dot{m}(\frac{P_1}{\rho_1} - \frac{P_2}{\rho_2}) = (\dot{m}\bar{c}_p)_C(T_{Co} - T_{Ci}) + \dot{m}(\frac{P_3}{\rho_3} - \frac{P_4}{\rho_4})$$ **Equation 3.11**

Where:

P_1, P_2 , P_3 and P_4=Pressure at connections 1, 2, 3 and 4 [Pa]

ρ_1, ρ_2, ρ_3 and ρ_4=Fluid density at connections 1, 2, 3 and 4 [kg/m^3]

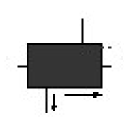

Figure 3.3 Thermal heat exchanger

3.2.3 Pumps

The component called *radial flow pump* (Figure 3.4) is used to model the operating characteristics of a rotodynamic pump. The pump component is provided with curves describing the pressure rise vs. flow rate and the torque vs. flow rate.

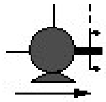

Figure 3.4 Radial flow pump

The energy input into the fluid is assumed to be equal to the energy 'lost' by the pump. The pump efficiency is defined by the following equation:

$$\eta = \frac{\Delta P \cdot Q}{M \cdot \omega} \qquad \textbf{Equation 3.12}$$

Where ΔP is the total pressure increase across the pump. The torque M is calculated from the provided curves.

3.2.4 Valves

In our model the *ball valve* component (Figure 3.5) is used. Different valves are simulated by defining a specific loss coefficient vs. position curve. The valve position is specified by ratio (0.0 = closed, 1.0 = fully open). For the position ratio 0.25 until 1.0 data from the valve manufacturer are implemented. For low ratios (ratio < 0.25) the Flowmaster supplied curve is used. The loss coefficient is defined in the same way as for the heat exchangers.

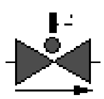

Figure 3.5 Ball valve

Heat cannot be lost or gained in this component.

3.2.5 Discrete loss

The *discrete loss* component (Figure 3.6) provides a way of modeling a pressure loss between two nodes where the only purpose of the component is to simulate a flow restriction within the system. The pressure loss can be defined by a fixed value of the loss coefficient, K, or by a curve that describes the variation of pressure loss with the flow ratio. The pressure loss can be specified dependent on flow direction.

Figure 3.6 Discrete loss

In the same way as for the valves, heat cannot be lost or gained in the *discrete loss* component. If the mass of the solid and fluid volume data are entered in the component data form, the energy balance on the component will include the effects of heat flow into or out of the material comprising the component and fluid thermal capacity.

3.2.6 Pipes

The *pipe* component (Figure 3.7) models the frictional pressure drop along a straight pipe assuming linear distribution of the frictional pressure gradient. The model assumes that the pipe has a constant cross-sectional area and a fully developed flow through the pipe.

Figure 3.7 Cylindrical pipe

The pressure drop in *bends* and *junctions* can be modeled by using discrete components (Figure 3.8).

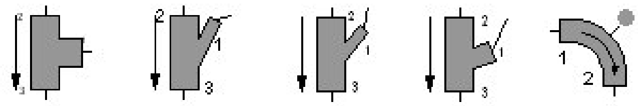

Figure 3.8 Discrete components represented by Bends and Junctions

The friction in the pipe depends on the pipe length, diameter and roughness. In the analysis the following equation describes the pressure drop:

$$P_2 - P_1 = \frac{fL}{d}\frac{\dot{m}_1|\dot{m}_1|}{2A^2\rho} \qquad \textbf{Equation 3.13}$$

Where f is the *Darcy friction factor*, L is the pipe length, d the pipe diameter and $\dot{m}_1$ the mass flow rate at arm 1. The non-dimensional loss coefficient is calculated as follows:

$$K = \frac{fL}{d} \qquad \textbf{Equation 3.14}$$

The friction factor is calculated by the Colebrook-White method. It combine a series of equations to model pressure drop in a pipe from laminar through transitional and into turbulent flow, applying the appropriate equation for the friction factor.

The expression that is used for Reynolds numbers less than or equal to 2000 is for laminar flow conditions:

$$f = f_l = \frac{64}{Re} \qquad \textbf{Equation 3.15}$$

where f_l is the Darcy friction factor for laminar flow.

For turbulent flow conditions, Reynolds number greater than 4000, the expression that is used to calculate the friction factor is:

$$f = f_t = \frac{0.25}{\left[\log\left(\frac{k}{3.7d} + \frac{5.74}{Re^{0.9}}\right)\right]^2} \qquad \textbf{Equation 3.16}$$

where f_t is the Darcy friction factor for turbulent flow and k is the roughness in [mm].

For transitional flow conditions, Re greater than 2000 and less than or equal to 4000, the following liner interpolation is applied from f_l to f_t:

$$f = xf_t + (1-x)f_l \qquad \textbf{Equation 3.17}$$

and

$$x = \frac{Re - 2000}{2000} \qquad \textbf{Equation 3.18}$$

There are seven options available for modeling heat flows from the liquid through the pipe wall to the surrounding ambient conditions. In our report the method for adiabatic flow is used. This method assumes no heat transfer from the liquid in the pipe. There will only be a small temperature rise in the flow direction caused by pipe friction.

3.2.7 Solids

There are seven components in this family. In the model the *heat flow source* and the *thermal bridge* (Figure 3.9) are used.

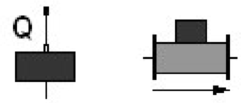

Figure 3.9 Heat flow source and Thermal bridge

Heat Flow Source

The *heat flow source* models the flow of heat to or from the solid part of the network. In our model it is used to implement the heat from the engine to the cooling system.

Thermal Bridge

The *thermal bridge* is modeled as a discrete loss with a contact area from which heat can flow. The component can be connected directly to two fluid nodes and one solid node to model the heat flow from the engine to the water/glycol mixture.

3.2.8 Generic component

As its name implies the *generic* component (Figure 3.10) can be used in a variety of ways. The component has no input data. Its functionality depends on the type of signal received from the connected *COM controller*. For example the component can be used to control volumetric flow rate, mass flow rate, pressure difference, humidity etc.

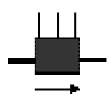

Figure 3.10 Generic component

In our model we use the *generic* component as a fan where the input signal is the required pressure difference.

3.2.9 COM Controllers

A *controller* imposes a signal upon a component. This signal can be read directly from user input data or derived from other data available in the network.

Gauge and Controller

Two common used *COM controllers* are the *gauge template* and the *controller template* components (Figure 3.11).

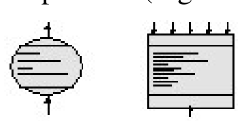

Figure 3.11 Gauge template and Controller template

Gauge template components have only a single measurement input connection. This may be connected to a node, a component branch, or to a component measurement output. The input signal is utilized in a script defined in the *gauge template* providing an output signal.

The output signal from a *gauge template* is often used as an input to a *controller template*. The *controller template* component can accept up to five inputs, which can be utilized through a script to provide an output signal.

Thermostat

The *thermostat* component is especially designed to handle transient heat transfer analysis, (Figure 3.12).

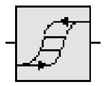

Figure 3.12 Thermostat component

The *thermostat* can model the hysteresis effect with two different curves, see Appendix E. This means that the *thermostat* has a different behavior when the temperature is decreasing compared to when the temperature is increasing.

Engine component and Vehicle component

The *engine* and *vehicle* components (Figure 3.13) are controllers implemented as pre-programmed *COM controllers template* with no input or output connections. The component stores a range of pre-defined data, curves and surfaces which can be accessed by *COM controllers* at all times in the analysis.

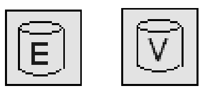

Figure 3.13 Engine and vehicle components

The components are designed to be used in vehicle management applications to control or set engine and driving cycle data, such as:

- Heat ,Q, varied as a function of RPM and Load.
- Vary vehicle velocity, engine speed and engine load with time.

Figure 3.14 shows how the components are used.

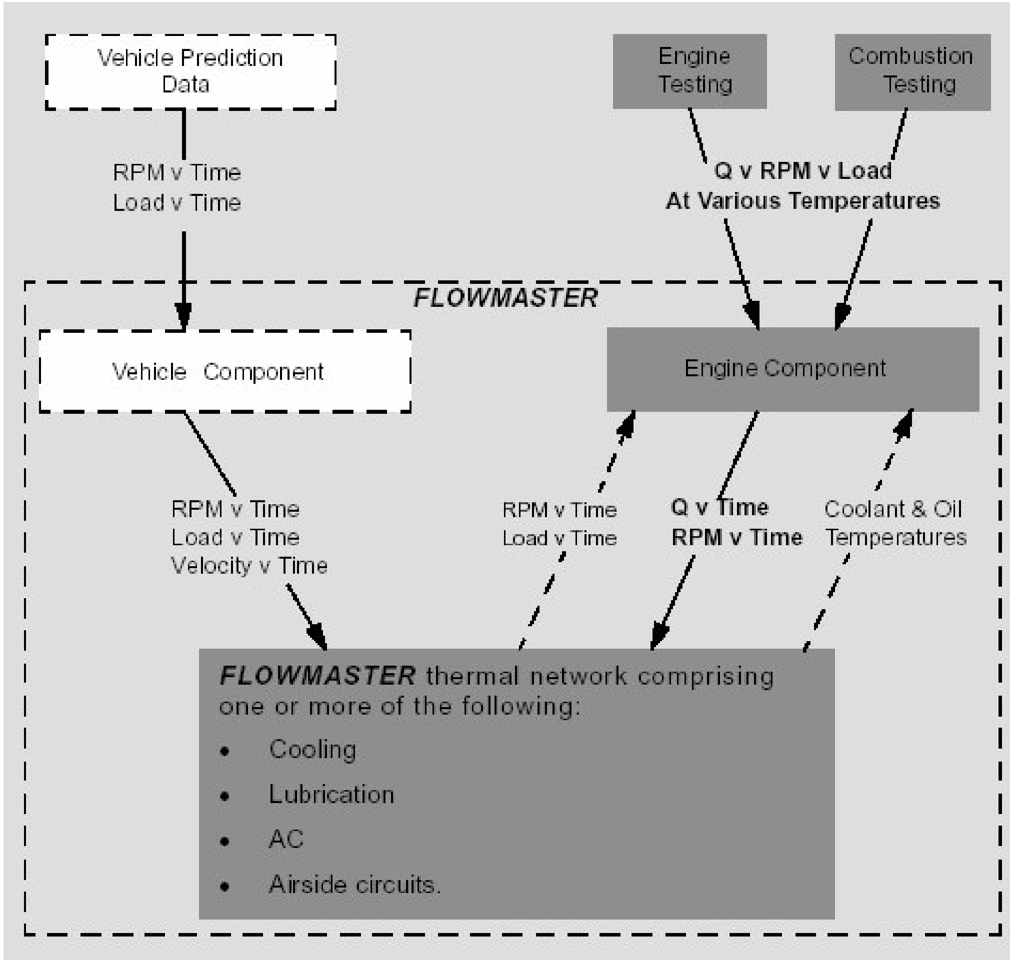

Figure 3.14 Example of Vehicle- and Engine Component usage

During an analysis, the vehicle component data and engine data is accessed by *COM controllers* which control the actions of other components in the network.

3.3 Validation

To be able to use the components in Flowmaster with actual component properties we analyzed the component data, obtained from the suppliers, and constructed tables and graphs for the components in Flowmaster. To validate the behavior of the emulated components, after inputting the characteristics of the physical components, we simulated each component by itself in forced conditions. The forced condition on flow rate, temperature and pressure makes it possible to compare each simulated component with the actual component data provided by the manufacturer. When all the included components were analyzed, single components was assembled to sub-systems and tested in order to control the effects of their cooperation. This approach clarifies the differences between the modeling tool and the actual tests. Finally, the complete network was assembled and evaluated.

4 Ram curve

To predict temperatures and flow rates in an engine cooling circuit, a large number of three dimensional CFD-analyses are typically required in order to determine the air flow patterns through the cooling package over a range of vehicle operating conditions. The primary aim of the 3D model is to calculate the air flow pattern through the bumper apertures and subsequently the cooling package. With such a model and enough CPU time, results similar to Figure 4.1 would be produced.

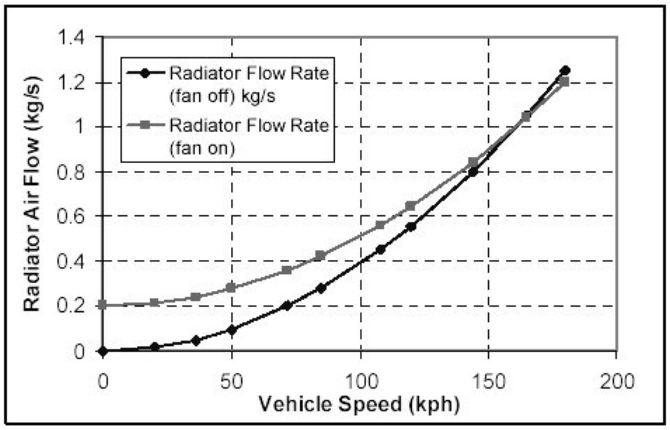

Figure 4.1 Radiator airflow results

These results could be used in a one-dimensional simulation tool to simulate the cooling package temperatures and airflow for one specific cooling package setup. If a component in the cooling package setup change for example in performance characteristics, a new set of CFD-solutions would be required. This procedure is very time consuming and requires a large number of CFD-simulations. To overcome this problem an alternative procedure is described called the *Ram curve concept*.

4.1 Theory

Calculations using the Ram curve and the Ram effect are mainly used in the aerospace industry where it is used in designing of the engine air inlet [8]. This approach eliminates the large number of time consuming CFD-simulations by characterizing the cooling package airflow performance and there by couple the 3D- with the 1D-simulation models.

If the airflow can be characterized, a large number of results can be produced using the rapid one-dimensional simulation tool based on relatively few CFD-simulations. This allows a change in the cooling package setup without a new set of CFD-solutions to be performed [4]. In order to characterize the airflow we analyze the pressure distribution over the underhood components.

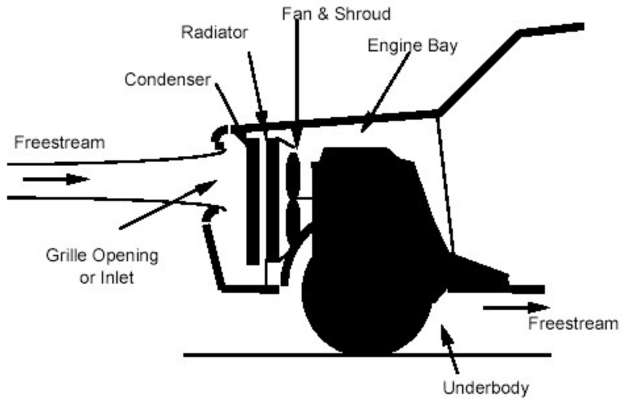

Figure 4.2 Vehicle cooling system

The cooling package described in this report principally consists of the components shown in Figure 4.2 and the pressure terms in the equations below are labeled in accordance to Figure 4.3.

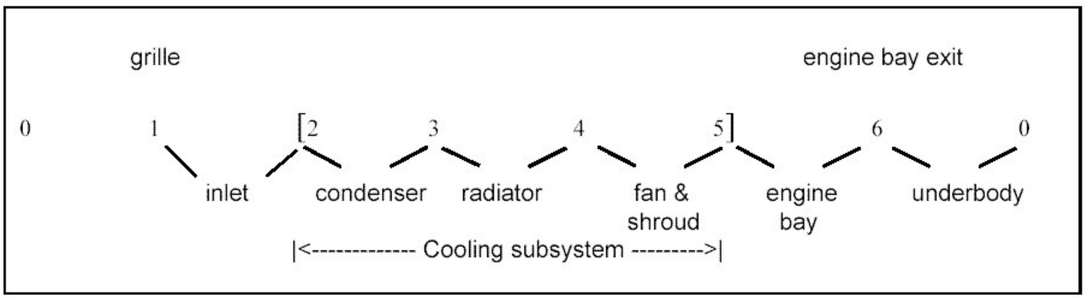

Figure 4.3 Cooling airflow path

The pressure difference over the cooling subsystem (cooling package) can be written as the sum of pressure losses and increases of the individual components [8]:

$$\begin{aligned} \Delta P &= (p_{t2} - p_5) = (p_2 + q_r) - p_5 = \\ &= (p_2 - p_3)_{con} + (p_3 - p_4)_{rad} - \left[(p_5 - (p_4 + q_r))_{fan+sh} + (\Delta p_i)_{CS}\right] \end{aligned} \quad \textbf{Equation 4.1}$$

Where $(\Delta p_i)_{CS}$ is the internal interference between the cooling package components, p_{t2} is the total pressure at the condenser face and q_r is the radiator airflow based dynamic pressure. The interference is mostly related to the fan and shroud design and the last term in the expression is for that reason called the installed-fan curve.

Since the pressure drop in a closed system must summate to zero, equation 4.1 can be written as the pressure variation through the inlet, engine bay and underbody. This expression is called the ram pressure and is defined as:

$$\begin{aligned} \Delta P_{ram} &= (p_{t2} - p_0)_{inlet} + (p_5 - p_6)_{bay} - (p_6 - p_0)_{ub} = ((p_2 + q_r) - p_0)_{inlet} \\ &+ (p_5 - p_6)_{bay} - (p_6 - p_0) \end{aligned} \quad \textbf{Equation 4.2}$$

The pressure loss in the engine bay region, $(p_5 - p_6)_{bay}$, is usually small in comparison to the under body (ub) pressure rise, $(p_6 - p_0)_{ub}$, and the pressure rise caused by the frontal projection of the inlet i.e. $(p_{t2} - p_0)_{inlet}$.

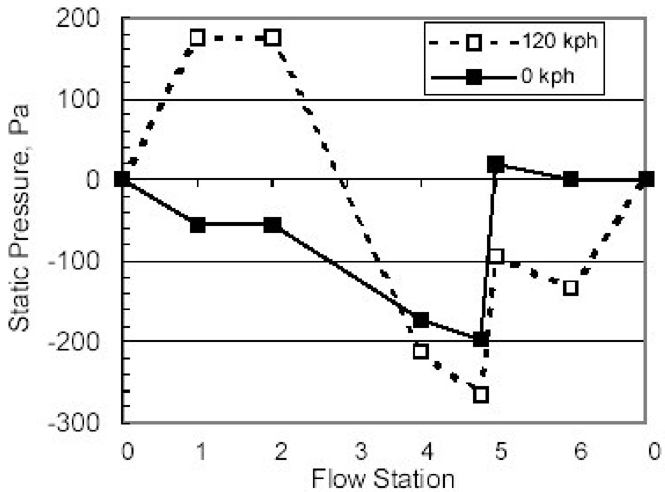

Figure 4.4 Cooling package pressure change

The ram pressure isolates the effect of exterior, engine bay and underbody sheet metal from the heat exchanger system (equation 4.2). Hence, it can be used as a measure of efficiency of the vehicle front end geometry (grille) in providing cooling airflow to the cooling package, independent of the cooling package setup and design including the fan speed. Figure 4.4 illustrates the effect on pressure at idle and 120 kph. When the vehicle is at idle and under the influence of the fan the inlet (grille opening) is a restriction to airflow and creates a depressed pressure level which the fan and ram have to work against. When the vehicle is moving however, the inlet recovers ram energy from the free stream to overcome this restriction and unload the fan.

To characterize the airflow we need a normalized expression of the pressure in order to account for the large variation in viscosity (consequently Reynolds number) with temperature.

$$K = \frac{\Delta P}{\frac{1}{2}\rho u^2} \qquad \textbf{Equation 4.3}$$

Equation 4.3 is often used to express the normalized pressure drop for a component in varying conditions, where K is the pressure loss coefficient and $1/2\rho u^2$ is the dynamic pressure for the fluid at interest. Analogously, the Ram pressure and radiator airflow is normalized as [4]:

$$K_r = \frac{\Delta P_{Ram}}{q_r};\ Q_{norm} = \frac{q_r}{q_0} \qquad \textbf{Equation 4.4}$$

Where q_r is the radiator airflow based dynamic pressure and q_0 the free stream dynamic pressure:

$$q_r = \frac{1}{2}\rho V_r^{\ 2};\ \ q_o = \frac{1}{2}\rho V_v^{\ 2} \qquad \textbf{Equation 4.5}$$

K_r is called the ram coefficient. The result of this normalization is a linear expression of air resistance as a function of vehicle speed.

$$K_r = a \cdot \frac{q_0}{q_r} + b \qquad \textbf{Equation 4.6}$$

Equation 4.6 generates the Ram curve.

4.2 Coupling of Three-dimensional and One-dimensional Simulation models

To receive the in-data values needed in order to derive the Ram curve, measurements are needed in form of actual wind tunnel tests or CFD-simulations. We choose to use data from CFD-simulations due to the lack of time of this project [9]. The results from the simulation were available only four days after the initiation of this work.

The simulations were conducted with four vehicle speeds for fan On/Off conditions which gives a total of eight independent simulations and produces eight sets of results.

The 3D model in the CFD-simulations was constructed in such a manner that no leakage flow was possible i.e. all the incoming air through the grille passed the underhood components.

According to the Ram curve theory discussed earlier the Ram curve should be independent of the cooling package pressure rise/drop and there for also the fan speed. To be able to verify the validation of this assumption the CFD-simulation was performed with two different fan speeds. Figure 4.5 illustrates a comparison between the two sets of data.

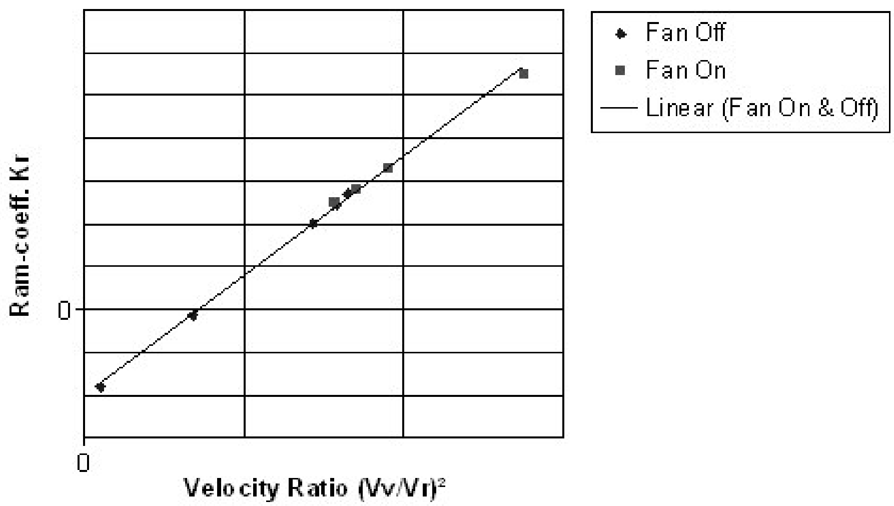

Figure 4.5 Illustration of the Ram curve

An evaluation of the normalized CFD-results for the fan on/off conditions shows that there is very little difference between the data sets and that the assumption regarding the independence of fan-speed is applicable. Hence the data can be aligned to a single Ram curve.

4.3 Deriving the ram curve

To derive the ram curve equations we used equation 4.4. From the CFD-simulations we retrieved the ram-pressure, ΔP_{ram}, and the cooling package air velocity, V_r. Using equation 4.5 and equation 4.6 the ram curve get the appearance of the linear function in Figure 4.5.

The negative intersection with the y-axis i.e. $K_r(0)$ is caused by the cooling inlet restriction at idle and the backpressure of the engine bay. The slope of the curve represents the influence of the efficiency of the grille openings and the uniformity of the flow [8]. For a model at ideal conditions the intersection is at the origin and the slope of the curve is 1.

4.4 Using the ram curve

When the ram curve was constructed one single vehicle velocity and a set of air-velocities was chosen. For every air velocity a ram pressure (head) was calculated using equation 4.4. These data forms a curve similar to a pump curve and represents one single vehicle velocity. To be able to use this pump curve for every vehicle velocities the curve has to be moved parallel using the similarity equations for pumps:

$$\frac{Q_1}{Q_2} == \frac{V_{v1}}{V_{v2}} \ (Flowrate)$$

$$\frac{H_1}{H_2} == \frac{V_{v1}^{\ 2}}{V_{v2}^{\ 2}} \ (Head)$$

Equation 4.7

These equations produce the results shown in Figure 4.6.

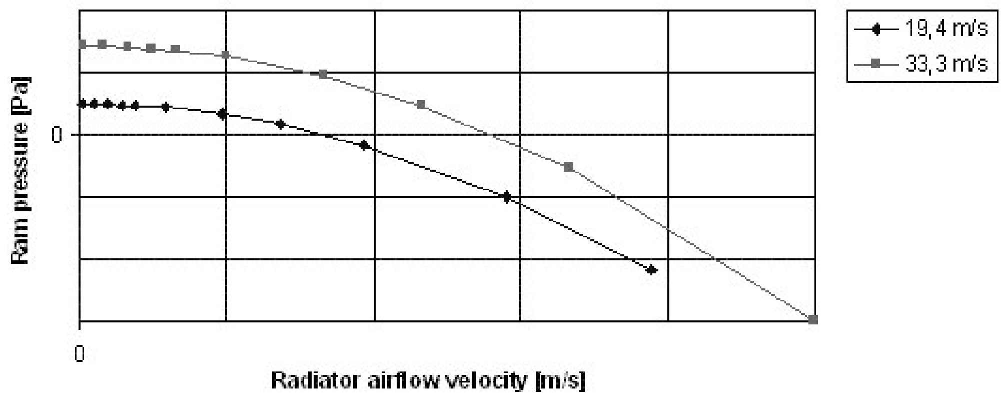

Figure 4.6 Ram curve for two vehicle speeds

5 The Cooling systems

When gasoline is combusted in an engine only 30 % of the chemical energy in the fuel is converted to useful work. The remaining energy is transformed to heat which must be removed from the engine in order to prevent the engine parts from overheating. To increase the heat rejection from the engine a cooling system is utilized in order to transfer generated heat to the surrounding air.

The cooling system in a modern car mainly consists of three major subsystems, separated by the difference in cooling fluid, which is directly or indirectly connected to the engine (Figure 5.1). These subsystems divert about 50 % of the generated heat in the engine and transfer it to the respective cooling fluid. The remaining heat is transferred from the engine mainly by the exhaust and by radiation and convection from the engine solids to the surrounding chassis and air.

In this thesis we used measurement data of absorbed energy by the coolant and oil at different driving conditions. These tests are simple to conduct in contrast to heat rejection measurements of the entire engine. As a consequence, heat rejection by radiation and convection of engine solid parts by the passing under hood air flow does not have to be treated.

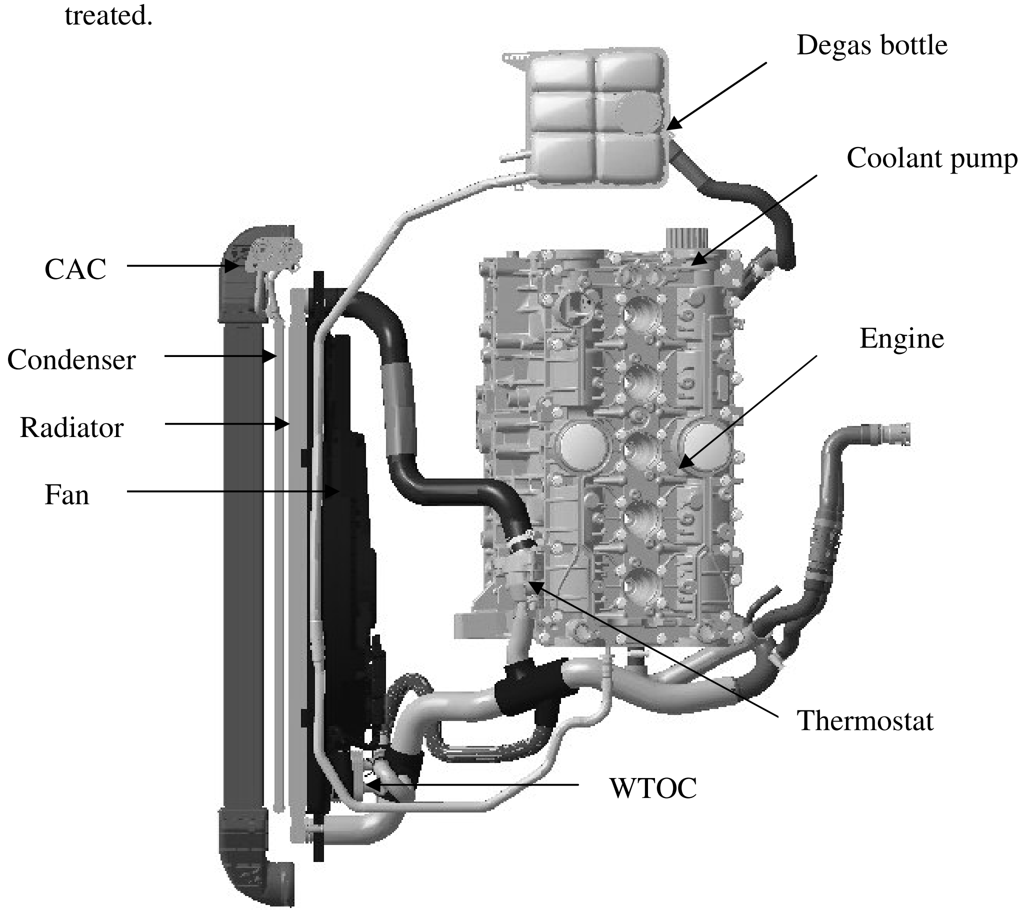

Figure 5.1 Coolant circuit

5.1 Engine

The engine treated in this thesis is a six-cylinder gasoline engine, modeled with and without a turbocharger, called SI6 T and SI6 NA respectively.

SI6 NA is a naturally aspirated model of the SI6 engine which means that the required air into the engine combustion is taken from the incoming air under the hood at atmospheric pressure and density.

The SI6 T is a turbocharged gasoline engine which means that the air used in the combustion is pre-compressed by a turbocharger, driven by the exhaust gases from the combustion. This results in a pressure rise and a larger quantity of the incoming air being forced into the engine hence more fuel, creating more power. Due to the larger amount of fuel injected per combustion the heat generation increases and consequently the engine requires enhanced cooling capacity.

An engine acts as a heat exchanger where the generated heat, originating from the combustion process and frictional forces in the engine, is transferred to the cooling systems. This is accomplished by circulating cooling fluids through passageways at the critical areas. For further information regarding the engine, see Appendix C.

5.2 Air cooling circuit

Due to the objective of this thesis, the focus of interest has been on the air cooling subsystem. All heat transferred from the engine to the subsystems are eventually diverted to the air cooling circuit, either by heat transfer in the radiator or convective air cooling of the engine block by the underhood airflow.

The air cooling subsystem mainly consists of the components mounted in front of the engine i.e. bumper/grille, condenser, radiator and fan. Air enters the vehicle through the grille and bumper setup and continues through the condenser/radiator cooling pack where the air is heated. The radiator is a heat exchanger where heat transfer between the coolant and air systems occurs. Due to the pressure drop in the mentioned components the airflow is low or even brought to standstill if the vehicle speed is below a critical value which creates a poor to non-existent cooling capacity. This trend is also identified at zero vehicle speed. To increase the airflow at low vehicle speeds a fan along with a shroud is mounted after the radiator which creates a lower pressure ahead of the fan and consequently increases the airflow. The fan speed is controlled by several parameters such as AC-pressure and temperature of the coolant at the radiator inlet.

For the turbocharged version of the engine an additional component is required in the cooling package, the Charge Air Cooler, CAC. This component is placed in front of the lower part of the condenser and operates as a heat exchanger between the hot compressed air from the turbo charger and the cooling air. The condenser and CAC components indirectly affects the heat transfer between the coolant and the air circuits in the radiator due to the fact that the cooling air passes through these components before it reaches the radiator. An illustration of the complete cooling package can be viewed in Figure 5.2.

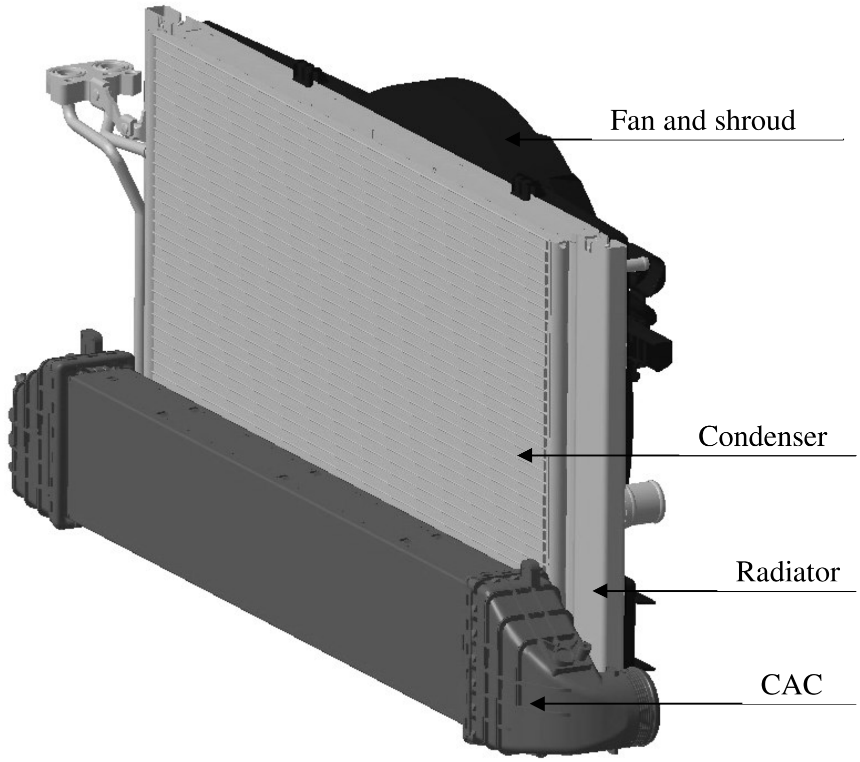

Figure 5.2 Engine air cooling package

Besides the effect on the mentioned components the underhood air flow furthermore cools the engine block, cylinder head and oil pan by absorbing heat radiation and removing heat from the surface by convective heat transfer.

5.3 Coolant circuit

The fluid used in the coolant system is most often a mixture of water and glycol (ethylene glycol) which increases the boiling point and lowers the freezing point compared with regular water. This allows the vehicle to operate in a wider temperature range.

The coolant enters the engine by a water pump and is circulated around the engine cylinders and through the cylinder head. At the engine coolant exit a thermostat is located. If the coolant is below a critical temperature the thermostat is closed and the fluid is directly recirculated to the engine inlet via the by-pass circuit. This is a very effective way to heat up the engine from a cold condition to operational temperature where the efficiency is higher. If the working temperature of the engine is reached (approximately 90 °C) and increasing the thermostats starts to open and coolant flow is partly diverted through hoses to the radiator circuit where the coolant is cooled by the radiator air flow. The water/glycol mix is transferred from the radiator outlet to the water pump before it re-enters the engine.

A part of the out-going coolant flow from the engine is redirected to the engine oil cooler, (EOC). This is a heat exchanger, connecting the engine oil cooling system with the coolant system. In this cycle heat is transferred to the coolant from the engine oil and then the coolant is returned to the engine inlet via the water pump.

The coolant system has different arrangements depending on gear box configurations, i.e. manual or automatic. The automatic gearbox must be cooled by the coolant due to the large quantity of heat generation in this kind of gearbox. The coolant is transferred to the gearbox by diverting the flow from the upper ten radiator tubes, the *subcooler*, to a transmission oil cooler, the *WTOC*. The pressure drop in the subcooler is higher than in the remaining part of the radiator due to additional resistance in the upper flow path. In the *WTOC* the hot transmission oil is cooled by the coolant and recirculated in to the gearbox.

To be able to control the cabin climate of the car, an individual circuit with a heat exchanger for the cabin air is installed. To this system a small part of the hot coolant from the cylinder head is diverted. In our model this heat exchanger is represented by a *discrete loss* component modeling the pressure loss in the heat exchanger. This is sufficient since we only consider the worst case scenario where the cabin heater is turned off.

5.4 Oil cooling circuit

The oil cooling circuit is a combined system where both cooling and lubrication is provided to the engine components and transmission. It is essential that the temperature of the oil is at an appropriate level due to temperature dependent changes of viscosity. The viscosity changes affects the oil film thickness hence also the lubrication ability.

Generally, the main critical areas in the engine which needs to be cooled and lubricated by the engine oil in order not to be overheated is the crank mechanism (including the piston arrangement) and valve train. The crank mechanism is exposed to heat caused by the combustion process and frictional induced stresses associated with the rapid motion of the parts (bearings, piston rings, crank shaft etc). The valve train is a collection of components controlling the opening and closing of the valves located on top of the cylinder head. For the valve train it is primarily the exhaust valves which need to be cooled as a result of the hot exhaust gases [6]

The oil is drawn from the oil pan by an oil pump and forced into an engine oil filter and cooler (EOC). After the oil has been cooled it is injected to a system of pipes and passageways connecting the oil with the vital engine parts where cooling is necessary. Finally, when the oil has passed the critical components via the passageways it returns to the pan and completes the cycle.

In the gearbox the transmission oil is circulated to lubricate and cool the gears in the same manner as the engine oil.

6 Components in the air cooling circuit

The setting of components in the air cooling circuit varies depending on if the engine is turbocharged or not, i.e. SI6 NA or SI6 T. Consequently they will be treated separately in the following chapters.

6.1 SI6 NA

Figure 6.1 illustrates the air cooling circuit in Flowmaster for the SI6 NA with two different gear box configurations. Depending on the gear box setup, the radiator is simulated in two separate ways.

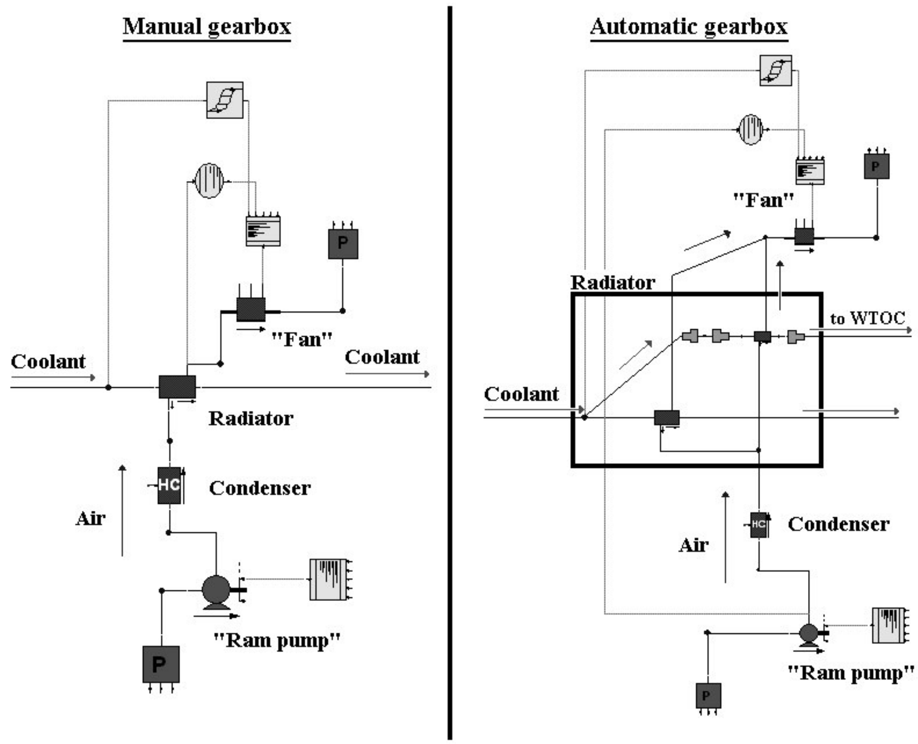

Figure 6.1 Air side cooling circuit for SI6 NA, representation in Flowmaster

6.1.1 Grille/Bumper

The air intake in the front of the car is called the grille. When the car is moving forward the static pressure of the intake air through the grille increases as a result of decreasing air velocity (Bernoulli's equation). The shape and size of the grille governs the appearance of the ram curve.

In the Flowmaster model the grille is represented by a pump controlled by the vehicle velocity, the *Ram Pump* (Figure 6.2). The pump is governed by a pump curve which is moved parallel to match the current vehicle velocity, see Figure 4.6. The present resistance in the circuit determines the delivered head by the ram pump hence also the air flow.

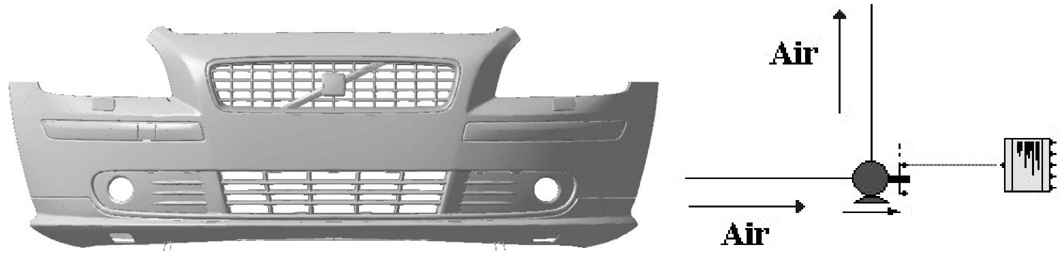

Figure 6.2 Grill/Bumper, representation in CAD and Flowmaster

6.1.2 Condenser

To achieve a pleasant cabin condition in the car it is equipped with an Air Conditioning system. The condenser is the main component in the AC-system. The cooling air temperature rise in the condenser will vary from case to case depending on the ambient temperature and the desired temperature in the car cabin.

In the model the condenser is simulated by a heat exchanger that delivers a constant amount of energy to the cooling air (Figure 6.3). This amount of energy represents the "worst" case where the AC is considerably utilized. The cooling air pressure drop over the condenser depends on the air mass flow and temperature. To be able to achieve the correct cooling air pressure drop, test data from the condenser manufacturers have been used.

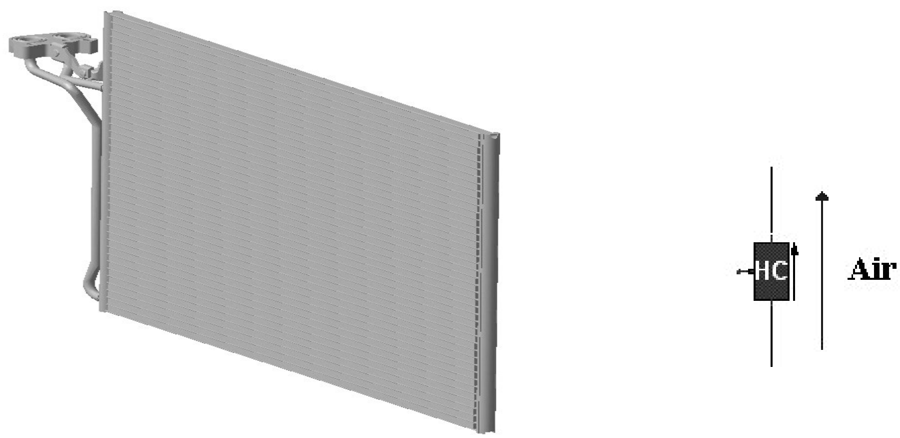

Figure 6.3 Condenser, represented in CAD and Flowmaster respectively

6.1.3 Radiator

The radiator is a component used to transfer heat from the coolant to the ambient air. The coolant flows in tubes from side to side, and the air flows in the transverse direction forced by the vehicle speed and under some circumstances by the fan. To increase the heat transfer, flat metal pieces known as fins are attached to the tubes. This increases the area where heat transfer can take place.

Depending on gearbox selection, two different configurations of the radiator exists.
In Flowmaster, the radiator for the engine with automatic gearbox has to be considered as two separate components. This is a consequence of the difference in pressure drop on the coolant side in the upper ten tubes (the *subcooler*) compared to the rest of the radiator. For the engine with manual gearbox the radiator has uniform properties and can be treated as one single component (Figure 6.4).

In the Flowmaster model the radiator is represented by a heat exchanger. Using the performance data from the radiator manufacturer, the heat exchanger components in Flowmaster is modified to perform as the real radiator with respect to heat transfer and pressure drop. For the engine with manual gearbox a single t*hermal heat exchanger* component represents the physical radiator in Flowmaster, and for the engine with automatic gearbox two separate *thermal* components are used.

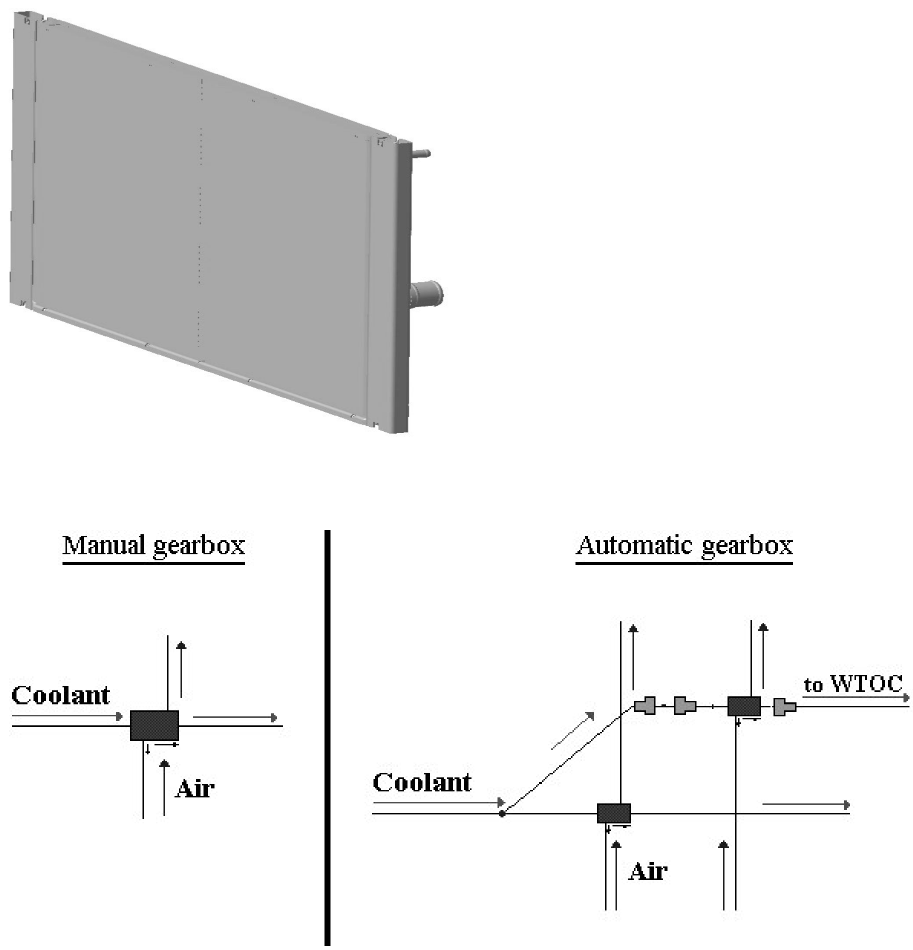

Figure 6.4 Radiator, represented in CAD and Flowmaster respectively

6.1.4 Fan

When the airflow, created by the vehicle speed, through the air cooling system components is to low, a fan is necessary to prevent the engine from overheating. The fan is placed behind the radiator. In the naturally aspirated engine a single fan is used due to the relatively uniform air flow and pressure drop over the cooling package, see Figure 6.5. It is an electric fan with five different speed steps. The fan switches between the steps depending on the coolant temperature, AC-pressure and several other parameters.

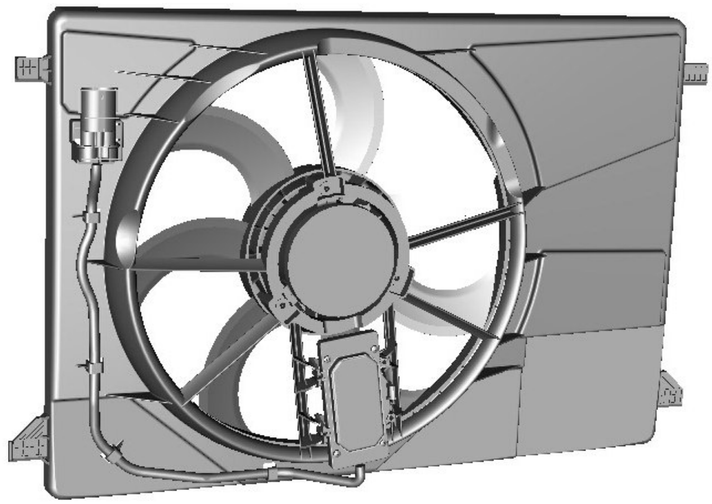

Figure 6.5 Electric Single Fan

In the Flowmaster model the cooling fan is represented by a generic component. It is governed by a thermostat, a gauge and a COM controller (Figure 6.6).

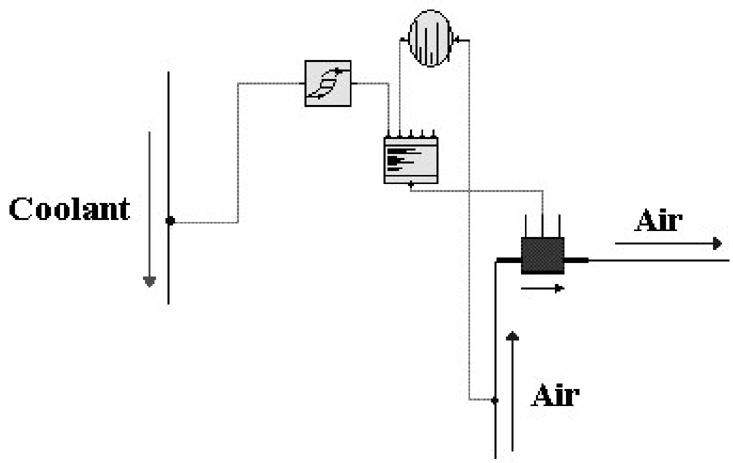

Figure 6.6 Fan, representation in Flowmaster

The thermostat measures the coolant temperature and delivers the proper fan step via two hysteresis curves to the controller (see Figure 6.7) [5]. The gauge component measures and delivers the air volume flow through the radiator. The controller contains five fan curves, see appendix K. When the fan step and air volume flow are set the controller utilizes the fan curves to determine and deliver the corresponding pressure rise/drop to the generic component.

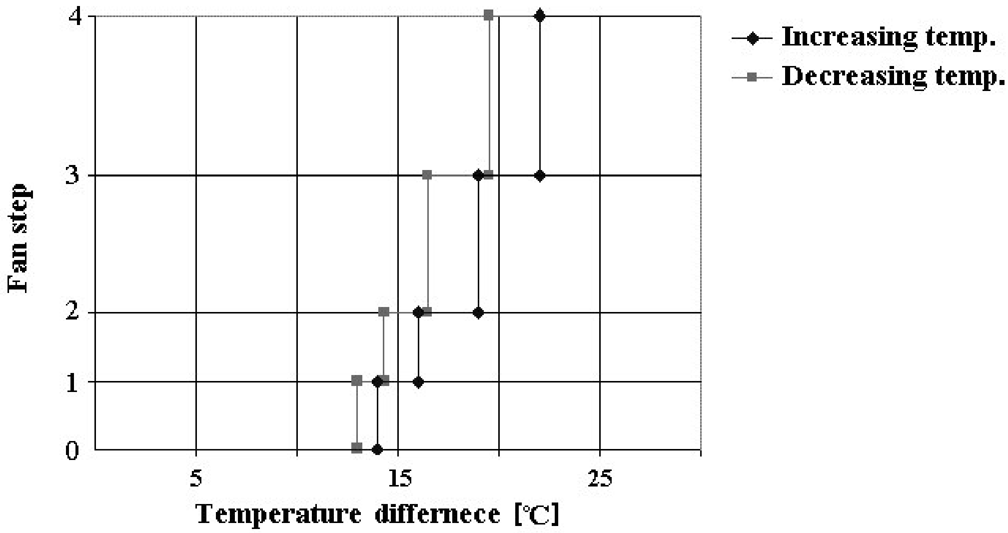

Figure 6.7 Fan step vs. coolant temperature

6.2 SI6 T

The cooling system component arrangement for the SI6 T differs from the SI6 NA setup due to the effects and requirements of the turbo charger and needs to be described separately. In Figure 5.2 the air cooling circuit is illustrated. Figure 6.8 is the corresponding circuit represented in Flowmaster.

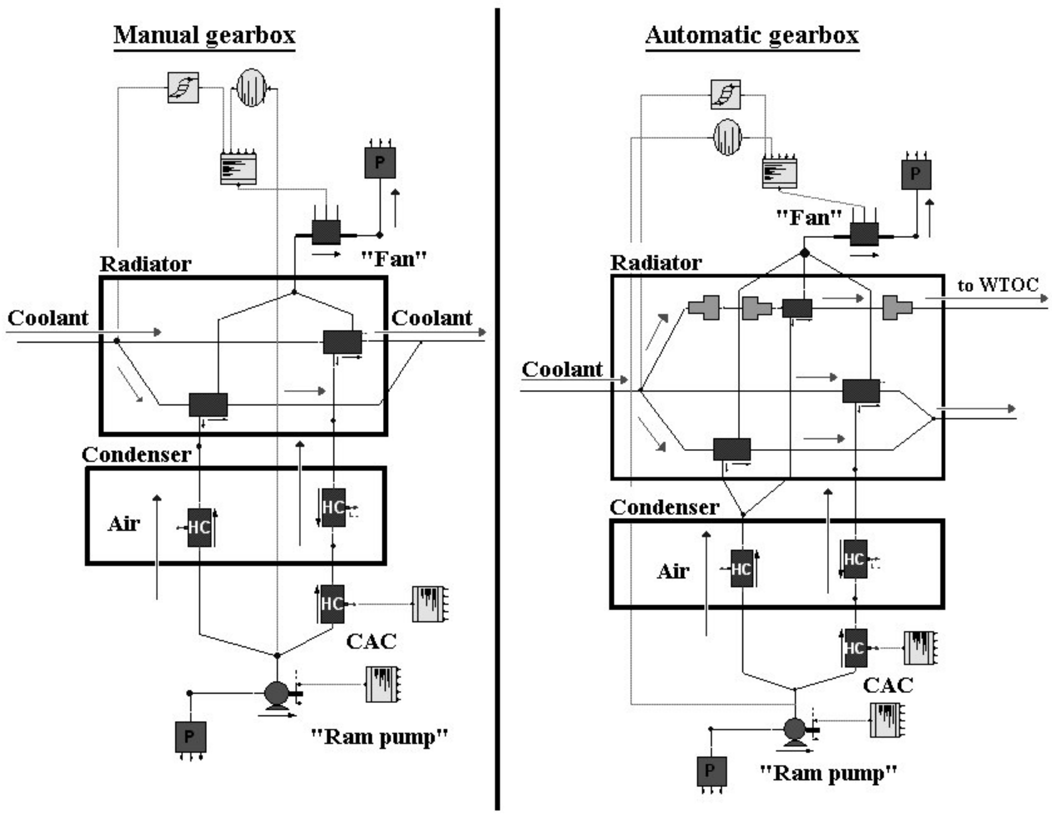

Figure 6.8 Air side cooling circuit for SI6 T

6.2.1 Grille/Bumper

The air intake for SI6 T is exactly the same as the SI6 NA setup.

6.2.2 Charge Air Cooler (CAC)

A turbo charger is normally limited by the high turbo outlet temperature, caused by the compression. To overcome this problem an additional component is included in the cooling system called the Charge Air Cooler or CAC (Figure 6.9) which cools the charging air from the turbo charger. The decrease in charging air temperature into cylinder results in a favorable lower density and allows the turbocharger to deliver a higher boost pressure without overheating [6]. A Charge Air Cooler is placed in front of the lower part of the cooling package and increases the air pressure drop over the component set. It also raises the air temperature in the radiator and condenser due to the additional heat rejection by the charge-air flow.

Figure 6.9 Charge Air Cooler, represented in CAD and Flowmaster respectively

In Flowmaster the CAC is simulated by a *heater-cooler* component controlled by a *controller template*. The controller determines the appropriate rejected heat by the CAC to the air stream via a 3D-map.

6.2.3 Condenser

The condenser for the SI6 T arrangement is identical to the one in SI6 NA, but in the Flowmaster model the condenser has to be split in two separate components to make simulations possible with a CAC present. Due to the existence of a charge air cooler in front of the condenser and radiator the flow differs between the upper and lower parts of the heat exchangers which has to be split in two separate fractions in order to simulate the setup correct, see Figure 6.10 .

To split the components some approximations has to be done. Dividing the condenser and radiator and treating the parts as two independent radiators eliminates the mixing flow perpendicular to the radiator surface. The component properties are scaled in the same manner as the size scaling.

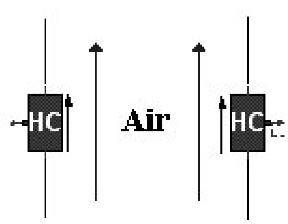

Figure 6.10 Condenser, representation in Flowmaster

6.2.4 Radiator

The physical radiator for the SI6 T is the same as for the SI6 NA. The difference is the way to model it in Flowmaster (Figure 6.11). Like the condenser the radiator is divided and scaled with respect to the size.

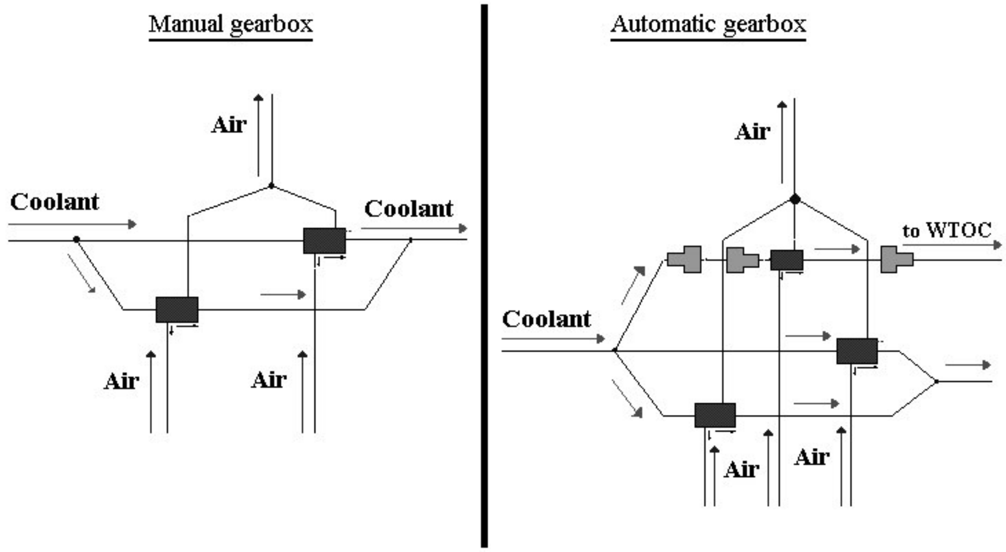

Figure 6.11 Radiator, representation in Flowmaster

In the same manner as the SI6 NA configuration, the radiator for the SI6 T is simulated in Flowmaster with two different radiator setups depending on gearbox selection.

6.2.5 Fan

Due to the additional temperature rise and pressure drop from the CAC the fan and shroud design looks different for the SI6 T than for the SI6 NA. Instead of one large single fan a two-fan design is used to maximise the cooling air velocity through the CAC and the lower part of the radiator and condenser (Figure 6.12).

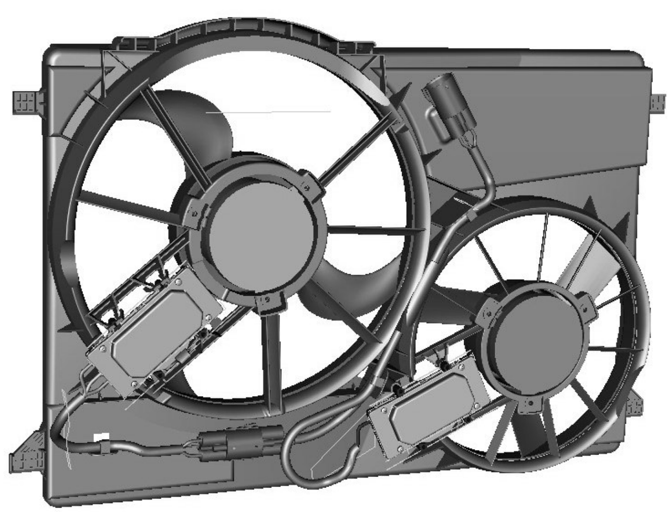

Figure 6.12 Electric Twin Fan

In Flowmaster the double fan is simulated in the same way as the single fan. The difference between the models is the setup of fan curves which determines the delivered pressure rise/drop Appendix K.

7 Approximations

All the components in Flowmaster are governed by equations, many of them approximated. For example, the system relies on simplified arrays of the Navier-Stokes equations even though the results become approximate.

In our model we have approximated the leakage over the cooling package to be zero. In the physical model a leakage flow over the heat exchangers can be observed which reduces the heat transfer.

The ram curve is a liner function created by approximating results from CFD-simulations, see Figure 4.5.

8 Results

The only available physical tests at this time were a wind tunnel setup of a car with a SI6 Turbo engine and an automatic gearbox. In the wind tunnel, three tests were performed: Hill climb with 1800 kg trailer (HCTR 1800), Hill climb with 2000 kg trailer (HCTR 2000) and Road drive cycle from 100 km/h to 180 km/h (RL 100-180). Parameters such as engine rpm, vehicle speed, torque and coolant temperature were logged during the wind tunnel test in a program called INCA. Using the engine/vehicle characteristic parameters obtained from these physical tests, these cases were also evaluated in our Flowmaster model of the SI6 Turbo engine with an automatic gearbox.

Due to the focus on the air cooling in this thesis, this chapter will mainly treat the components in the air cooling circuit. Other Flowmaster models can be viewed in Appendix A to J.

8.1 Evaluation of the Flowmaster model

To be able to match our model with the physical tests some modifications had to be done in our Flowmaster model. The coolant flow in the radiator circuit did not match with the results from the physical model. This is likely a cause of performed modifications of the physical water pump characteristics or pressure losses in cooling circuit components without updating the component data used in our Flowmaster model. These modifications has shifted the distribution of coolant flow in the system, hence also the cooling capacity. To compensate for the lower coolant flow through the radiator in the Flowmaster model we increased the pump gear ratio from the engine rpm from 1.39 to 1.65.

A total mass of 200 kg has been added to components in the Flowmaster model to emulate the effects of mass associated with components in the cooling circuit and engine block. The contribution of mass in the system has affects on the development of coolant temperature such as smaller temperature gradients and delayed coolant temperature response.

8.2 HCTR 1800

The HCTR 1800 case is divided into two separate phases, conditioning and hill climb. During the conditioning phase the vehicle with attached trailer is driven on flat ground with constant speed, rpm and load. This stage is a procedure to acquire appropriate working conditions for the engine such as temperature of coolant etc. After a certain time the coolant temperature has reached a steady value and the hill climb phase is initiated where the retracting force is increased. During the hill climb the vehicle speed is maintained at 70 km/h while the engine load and engine rpm is increased. The HCTR 1800 case is specified by the engine/vehicle characteristic parameters in Table 8.1.

	Conditioning	**Hill Climb**
Ambient Temperature [°C]	28	28
Vehicle Speed [km/h]	70	70
Engine Speed [RPM]	2000	3100
Engine Load [Nm]	175	260

Table 8.1 HCTR1800 engine/vehicle parameters

These parameters were inserted in our Flowmaster model which produced the following results:

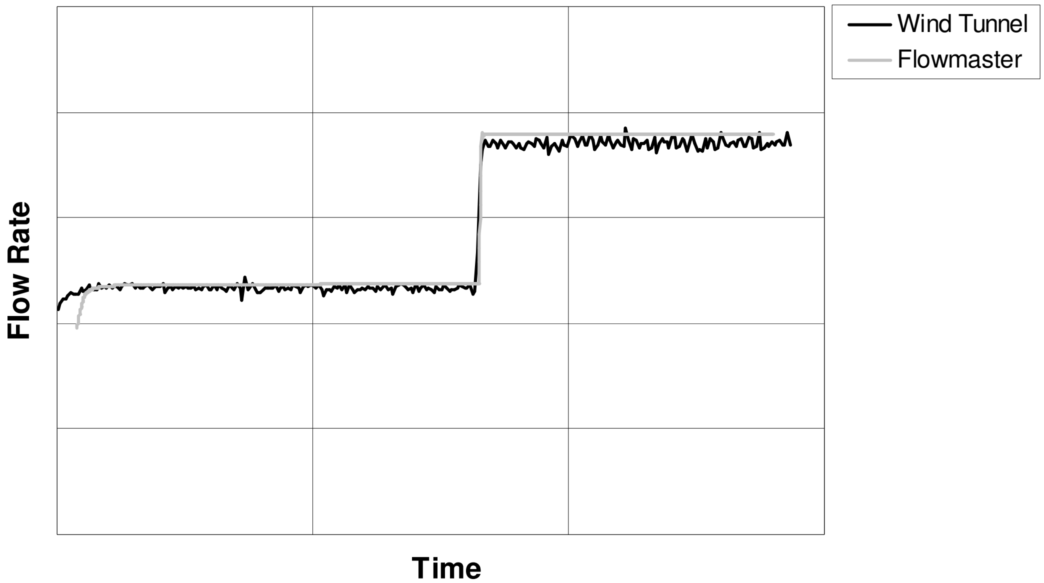

Figure 8.1 Coolant flow through radiator

After the modifications the coolant flow in the Flowmaster model (see Figure 8.1) coincide relatively well with the test results from the wind tunnel. This is essential for further comparisons between the model and the test results.

Figure 8.2 below illustrates the coolant temperature from our Flowmaster model compared with results from measurements in the wind tunnel.

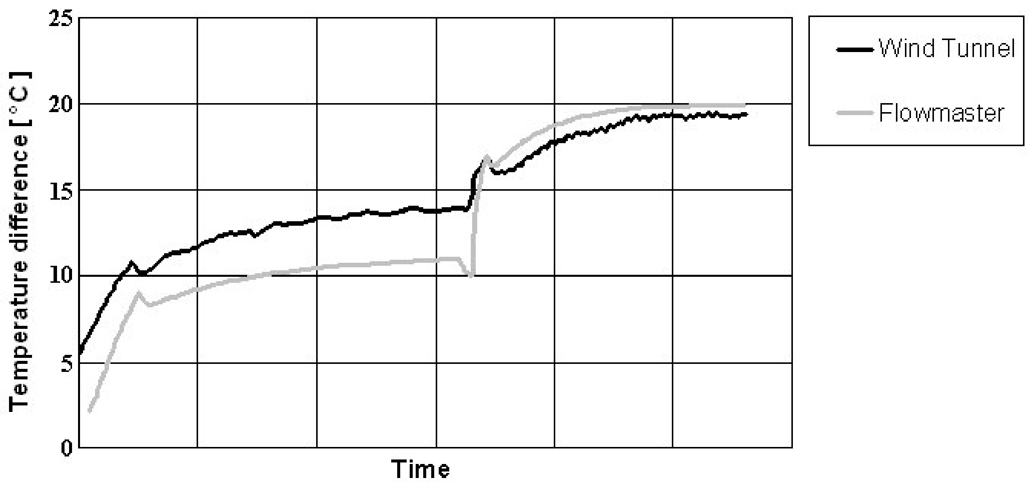

Figure 8.2 Coolant Temperature before radiator

The result deviation may depend on several parameters including inaccuracy of the ram curve or modification of physical components such as engine performance and fan performance etc. A possible error during the hill climb could be the altered coolant pump performance. When the thermostat is fully open the by-pass valve is closed. Due to the improved pump performance in our model, more coolant is forced through the engine oil circuit than regularly, generating a higher final coolant temperature than the physical test.

Other parameters are analyzed in the sensitivity analysis.

8.2.1 Sensitivity analysis

A powerful tool during simulations is the use of a sensitivity analysis. This is performed to evaluate the influence of changes in different parameters on the result and to be able to explain deviations.

Fan

The fan curves used in our model does not correspond to the actual behavior of the fan as a result of modifications of the fan steps and rpm during the physical tests. These modifications are performed to avoid problems such as disturbing sound and self-oscillation of the fan and shroud. To investigate the effects of fan characteristics on the Flowmaster model the performance of the fourth fan step was increased by 5 %. Modifications of fan step 1-3 did not influence the coolant temperature deviation during the conditioning phase considerably and was consequently not included in the sensitivity analysis.

In Figure 8.3 the difference in coolant temperature after the modifications in the fan characteristics is shown. This modification gives a slightly lower maximum coolant temperature which agrees even better with the wind tunnel results.

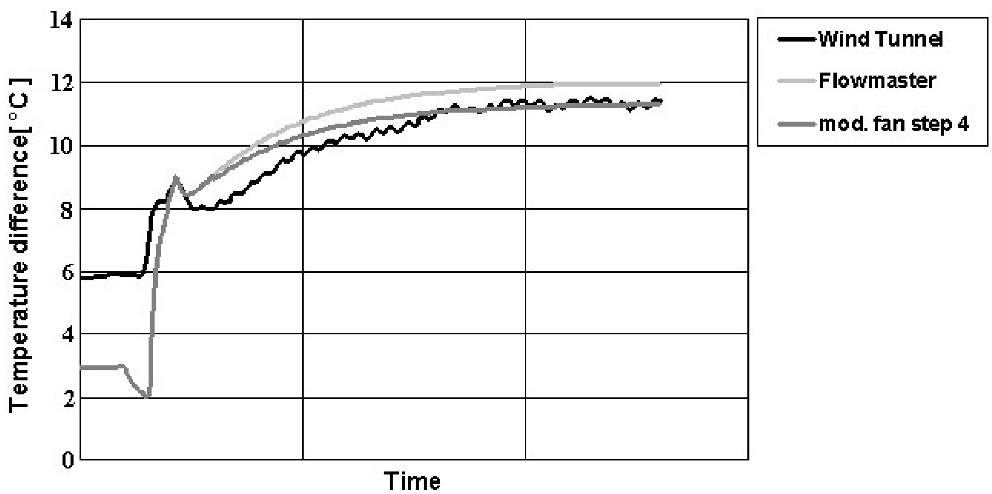

Figure 8.3 Coolant temperature before radiator

Mass

To evaluate the effects of component mass in the Flowmaster model we studied two cases with altered masses. As can be seen in Figure 8.4 the amount of mass has a great influence on coolant temperature behavior. In the case with no mass, coolant temperature is extremely sensitive to variations in fan step and driving conditions. For the case with a total component mass of 400 kg we achieve the opposite behavior with a sluggish temperature development. Observe that the influence of mass does not affect the steady state coolant temperature.

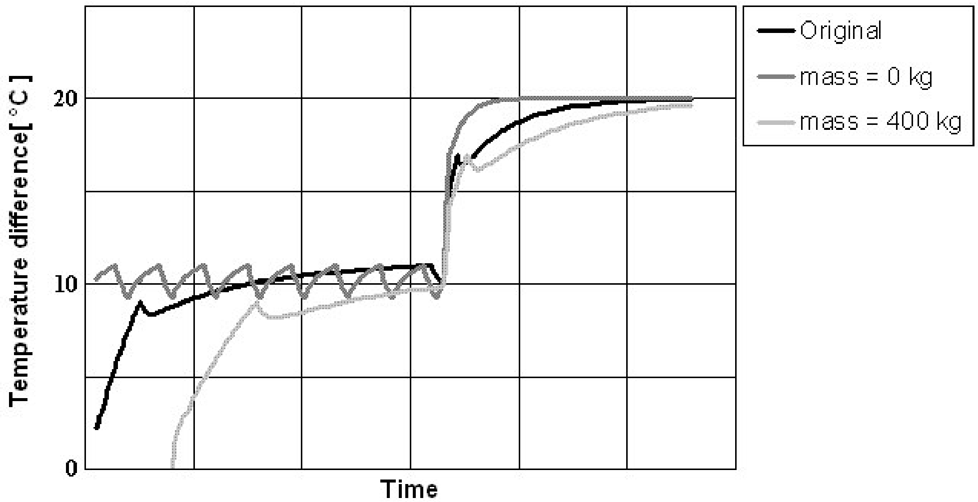

Figure 8.4 Coolant temperature before radiator

Time step

For transient calculations the time step is an important factor. If the time step is to big the generated solution might be inaccurate or even not able to converge at all. A decreased time step improves the accuracy of the solution at the expense of increased simulation time. It is essential to investigate if the time step used in the transient calculations is an appropriate balance between accuracy and simulation time.

Originally the HCTR1800 case was conducted with a time step of 1 second. This simulation required about 5 minutes to complete. When a time step of 2 seconds was used the solution did not converge. With a time step of 0.02 seconds the generated solution was more or less identical to the original solution. The simulation time for this time step required 2 hours to complete. Due to the insignificant gains in accuracy this time step is not recommended. This proves that a time step of 1 second is a suitable assumption.

Thermostat

During our work we have observed that variations of the thermostat performance have a major influence on the coolant temperature development. To investigate this behavior the performance was altered by increasing the opening curve (Appendix E) by 7 °C. These alterations produced the results shown in Figure 8.5.

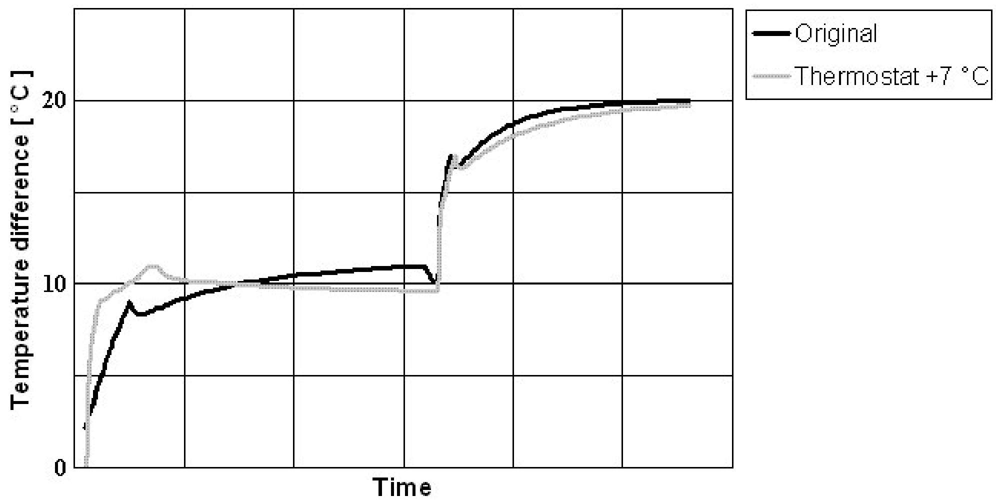

Figure 8.5 Coolant temperature before radiator

From the results we can see that the modified thermostat opens at a later stage which initially produces a higher coolant temperature. Due to the increased coolant temperature the fan changes fan step which causes the sudden temperature drop during the conditioning phase.

8.3 HCTR 2000

The HCTR2000 is a hill climb test with a 2000 kg trailer. When the conditioning phase is complete the hill climb phase is initiated with a maintained velocity of 70 km/h. The engine/vehicle characteristic parameters are illustrated in Table 8.2.

	Conditioning	Hill Climb
Ambient Temperature [°C]	28	28
Vehicle Speed [km/h]	70	70
Engine Speed [RPM]	2100	3150
Engine Load [Nm]	175	275

Table 8.2 HCTR 2000 engine/vehicle parameters

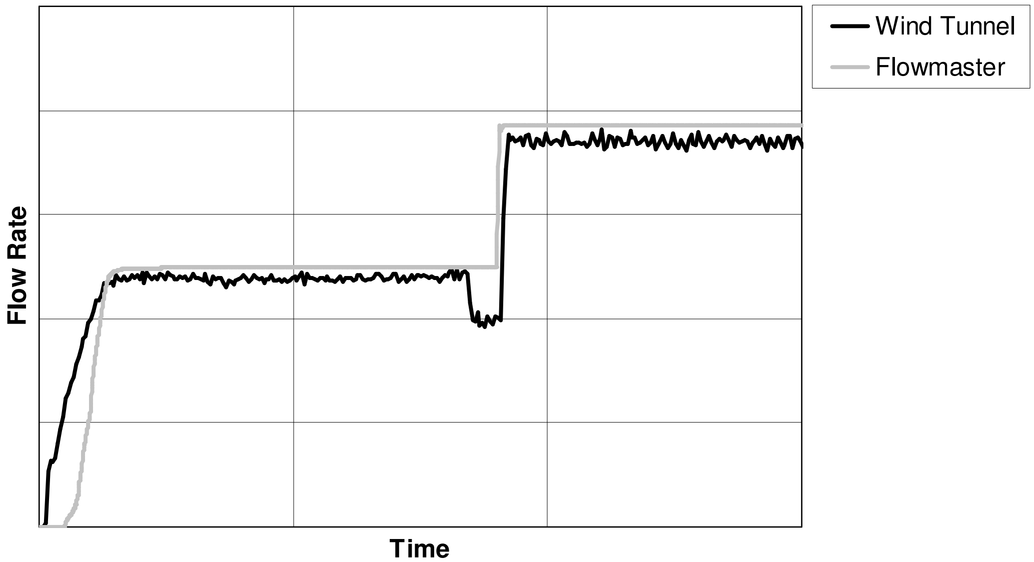

Figure 8.6 Coolant flow through radiator

Figure 8.6 proves that a pump ratio of 1.65 is an appropriate assumption also for this case.

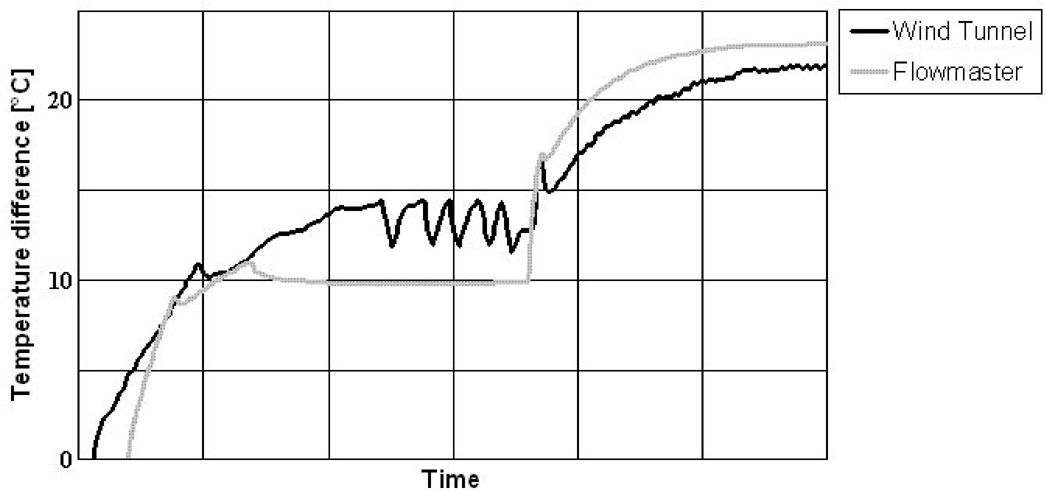

Figure 8.7 Coolant temperature before radiator

Figure 8.7 illustrates the coolant temperature. During both the conditioning phase and hill climb phase the coolant temperature shows the same tendencies as the HCTR 1800 case. The same explanations are applied on the cause of the deviations from the physical test. The oscillating behavior of the coolant temperature in the wind tunnel test during the conditioning phase is the result of alternations between two fan steps at that particular temperature.

8.3.1 Sensitivity analysis

Fan

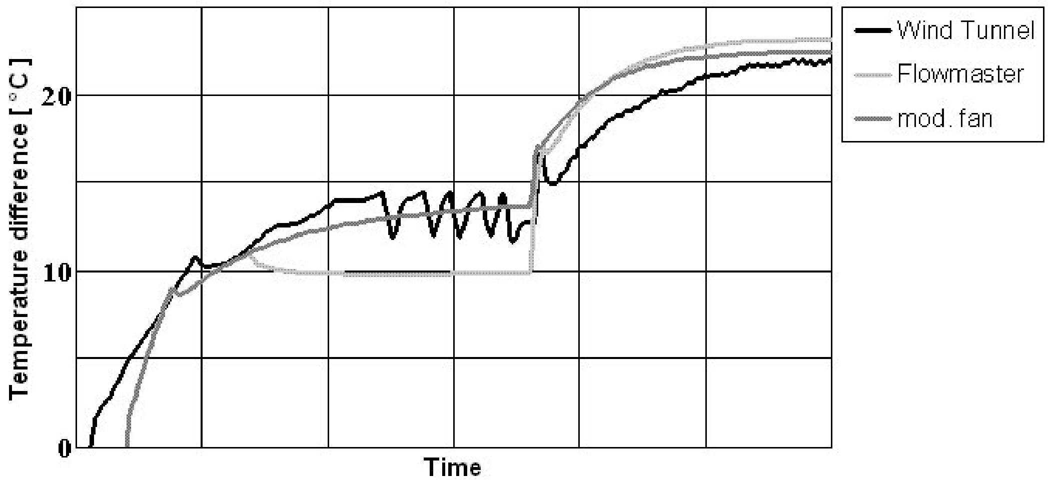

Figure 8.8 Coolant temperature before radiator

Just like the HCTR 1800 case the fan was modified by decreasing the last fan step (step 4) by increasing its efficiency by 5 %. Additionally we also changed the efficiency of the second step by decreasing its efficiency. In contrast to HCTR 1800, the last mentioned fan modification contributed to a better result during the conditioning phase, see Figure 8.8.

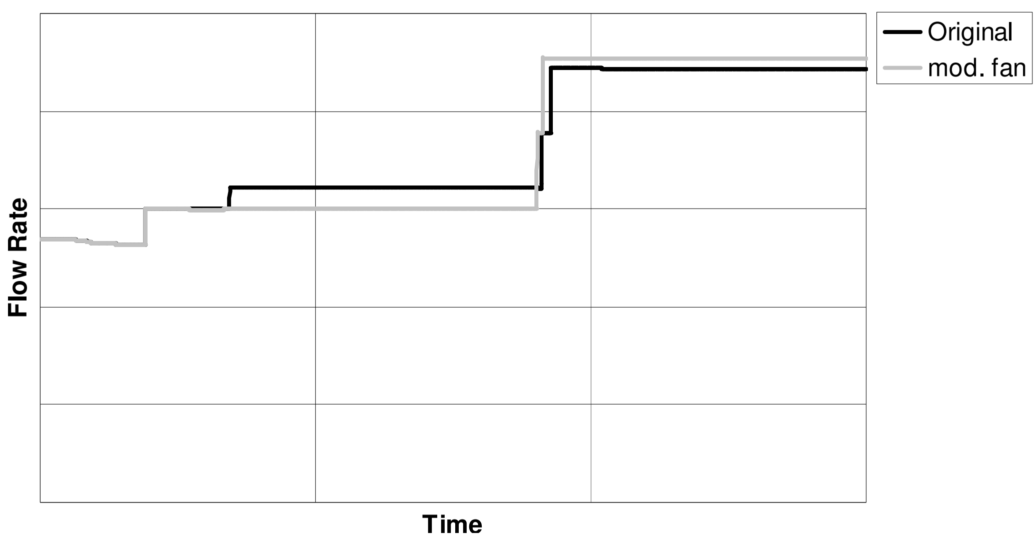

Figure 8.9 Radiator air flow

Figure 8.8 and Figure 8.9 illustrates the effects of air flow on coolant temperature. During the conditioning phase the modified fan generates a 6% lower air flow than the original simulation. The deviation in air flow creates a coolant temperature increase of 4 ℃ which proves that the coolant temperature is sensitive to changes in air flow.

Mass
Again, the influence of mass affects the results similar to the previous case (see Figure 8.10).

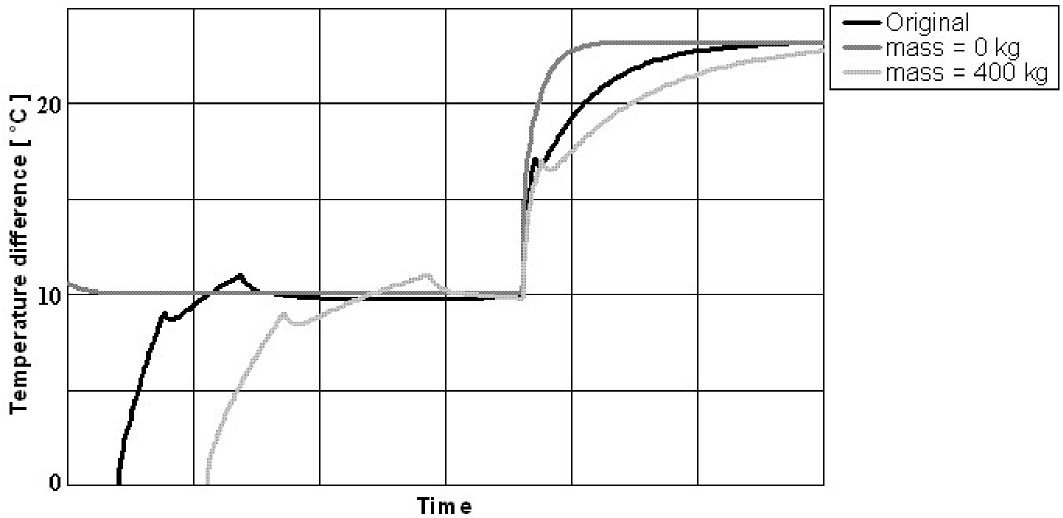

Figure 8.10 Coolant temperature before radiator

Time step
In the same manner as for the HCTR 1800 case the solution was analyzed by changing the time step. The study generated approximately the same results for the mentioned time steps and we concluded that a time step of 1 second is appropriate for the case in question.

Thermostat
Figure 8.11 shows the effects of a modified thermostat. Generally the temperature is higher than the original simulation. During the hill climb the temperature is slightly lower for the simulation with a modified thermostat. This is most likely caused by a lower coolant flow through the EOC, which leads to a smaller amount of transferred energy to the coolant in the heat exchanger. The distribution of coolant flow is determined by the differences in resistance between the by-pass loop and the engine oil circuit.

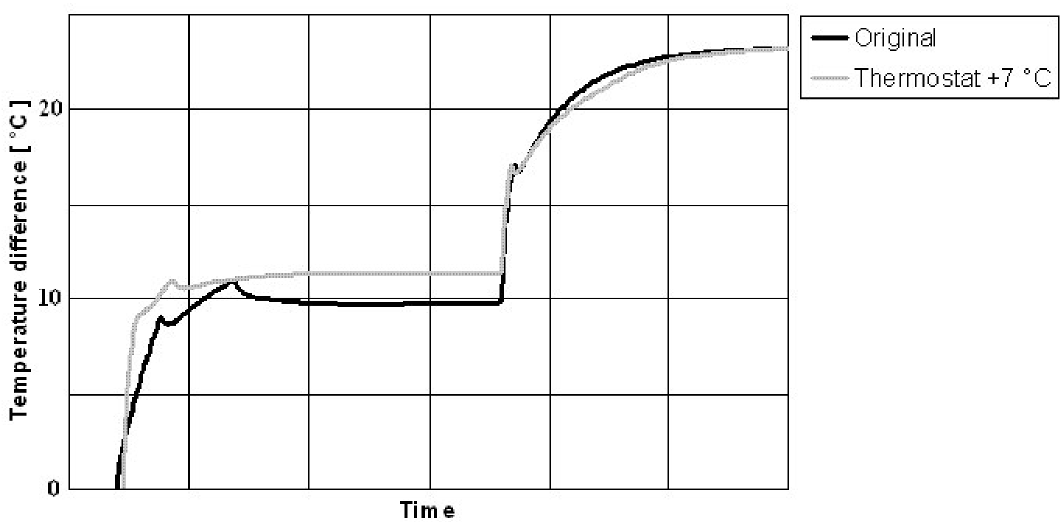

Figure 8.11 Coolant temperature before radiator

8.4 RL 100-180

The case called RL 100-180 simulates the car driving on a flat road with a vehicle speed increasing from 100 to 180 km/h. As can be seen in Table 8.3 below the case is divided into four parts with variations in vehicle speed. The ambient temperature is 38°C during the driving cycle.

	Part 1	Part 2	Part 3	Part 4
Ambient Temperature [°C]	38	38	38	38
Vehicle Speed [km/h]	100	140	160	180
Engine Speed [RPM]	1955	2765	3138	3517
Engine Load [Nm]	130	170	200	223

Table 8.3 RL 100-180 engine/vehicle parameters

Figure 8.12 illustrates the coolant flow through the radiator.

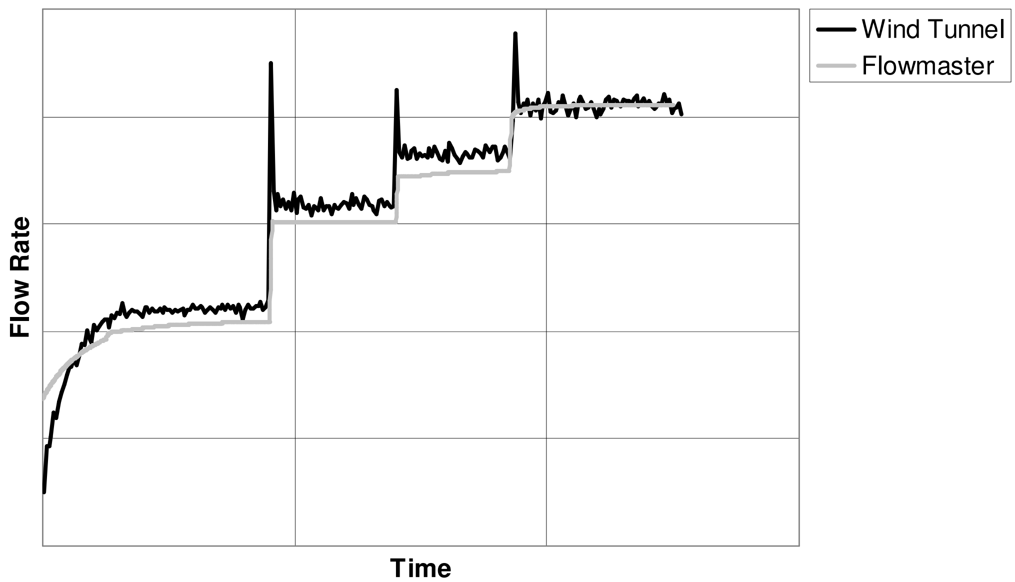

Figure 8.12 Coolant flow through radiator

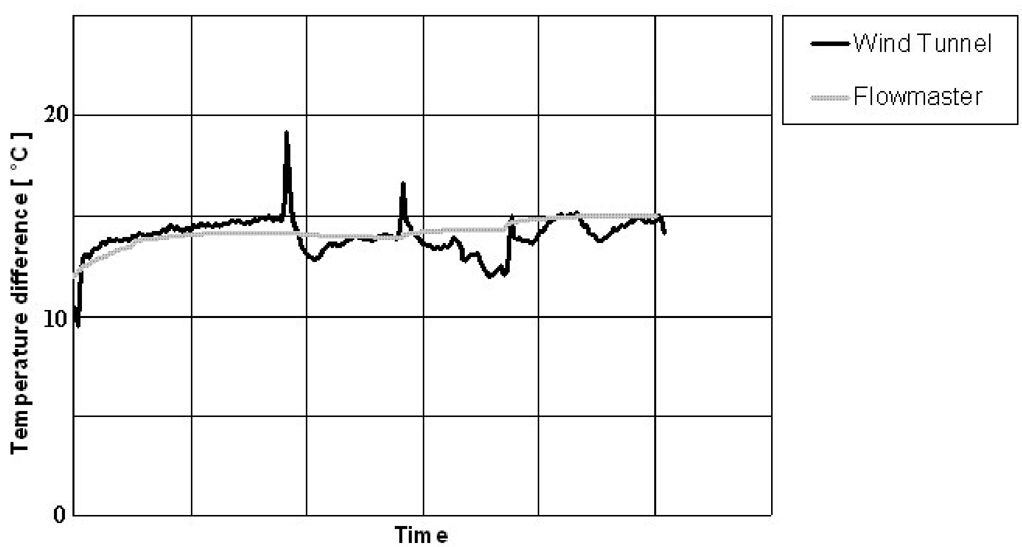

Figure 8.13 Coolant temperature before radiator

A comparison between the coolant temperature before the radiator in the Flowmaster model and the wind tunnel test is shown in Figure 8.13. The coolant temperature remains comparatively low through out the test because of a relatively low engine load and a high vehicle speed.

8.4.1 Sensitivity analysis

Fan

Due to the low coolant temperature, the fan is co rotating during the entire test, hence modifications of the active fan steps will not affect the result.

Mass

As expected, a greater component mass generates an initially lower coolant temperature. After a certain time the progress of the coolant temperature follows the same path as in the original Flowmaster simulation (see Figure 8.14). When a component mass of 0 kg was implemented, the solution did not converge. This is the result of inadequate damping and consequently a sensitive system.

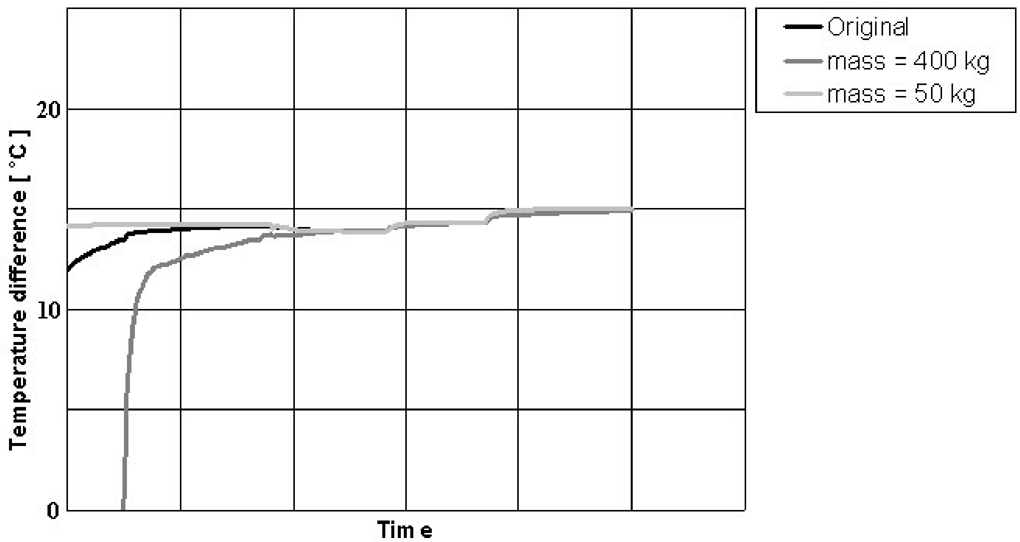

Figure 8.14 Coolant temperature before radiator

With a mass of 50 kg the simulation converges and produces a result with the same behaviour as the previous cases where the component mass has been small or non-existent.

Time step

For the analysis in this case a time step of 0.25 seconds is used. For larger time steps the solution did not converge, and with a time step of 0.1 seconds the generated solution was more or less identical to the original solution.

Thermostat

As we have seen in the pictures above the coolant temperature during the RL 100-180 case is comparatively low. This leads to a never fully open thermostat during the driving case and a modification of the curves describing the thermostat opening ratio vs. temperature has a large influence on the coolant temperature development. This is shown in Figure 8.15.

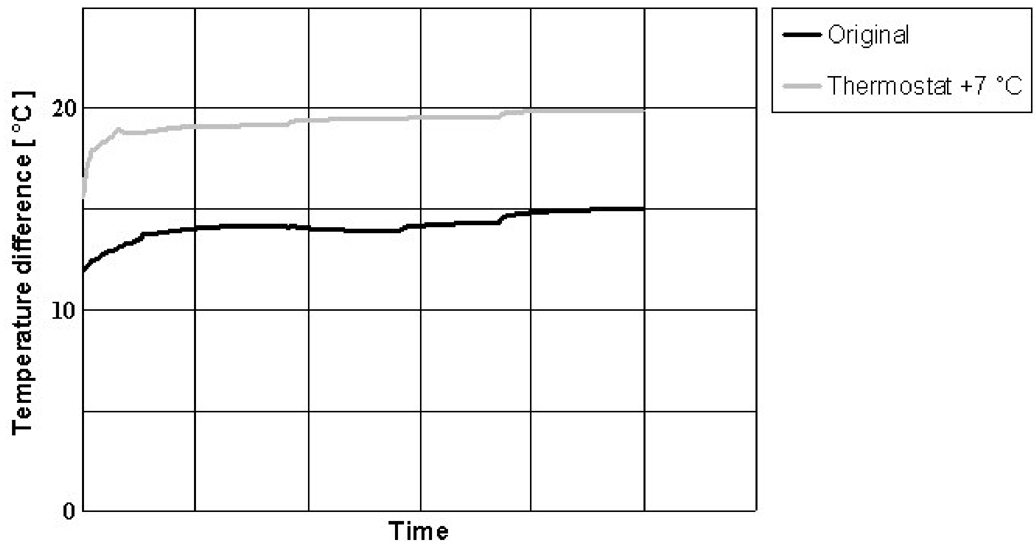

Figure 8.15 Coolant temperature before radiator

8.5 Source of errors

The 3D-engine maps which control the rejected heat to the coolant (Appendix C) were constructed by linearizing data points by a polynomial function. The linearization makes the data continuous but creates an error during the process. This deviation may be a cause of the sometimes diverging results. To overcome this problem the performed engine tests has to be extended to include more data points. This would generate a more accurate polynomial function.

When we obtained the engine/vehicle characteristic parameters from INCA the measurements had to be averaged to be able to use them in Flowmaster. In reality the parameters in, for example Table 8.1 are constantly changing and deviate slightly from the mean value. This averaging was necessary to obtain a converging result with a reasonable simulation time. To overcome this problem more data points from the INCA-measurements could be included at the expense of increasing simulation time and an enlarged risk of diverging results.

In Chapter 8.1 we mentioned that several physical components probably have gone through some modifications lately which causes the simulation results to differ from the physical tests, for example the water pump and resistance in the engine oil circuit. This is most likely a large source of error due to the large impact of coolant flow on coolant temperature. Another component that may have been modified is the engine. A modification of the engine performance affects the rejected heat to the system and consequently also the 3D-engine maps. This problem could easily be overcomed with improved communication between the construction and simulation departments.

As we mentioned earlier the ram curve was created by approximating the normalized CFD-results with a linearized curve. A small deviation from the linear ram curve creates a large divergence in air flow and consequently also the coolant temperature. As we know it, the ram curve technique is the only way to characterize the airflow without performing a new set of CFD-simulations at every case.

9 Recommendations and Conclusions

The produced results in Flowmaster show a relatively good congruity with the wind tunnel tests in spite of the approximations and sources of errors. Even tough approximations were made to construct the ram curve, the ram curve concept proved to be a powerful tool in predicting the radiator air flow. Since the Flowmaster model previous to this work is currently used in the development of new engines, we think that our new improved model can be implemented and used for even more complex problems. In spite of the congruity with the wind tunnel tests, further optimization of the Flowmaster model could be necessary to minimize the required simulation time and improve the results.

The next step in the development of the network might seem to be an implementation of the oil system in the Flowmaster model. A disadvantage with an implementation is the increasing simulation time with a growing network. It is important to remember that one of the main advantages with the 1D-simulation compared with the 3D-simulation is the rapid simulation speed. If the simulation time increases the benefits with the 1D-simulations decreases. Because of this it might be wise to have a separate system for the oil circuit after all.

We used a *generic* component to emulate the physical fan and consequently have total control on the required and produced pressure drop/rise. If a *fan* component should be implemented in the Flowmaster model further investigation would be necessary to match the physical component with the emulated.

To get a better congruence when the Flowmaster model is compared with the actual tests the 1D and 3D model should be expanded and include a leakage flow, divergent from the air flow path through the air cooling components. This necessitates further physical tests and analyses of the underhood air flow.

To be able to simulate the transient heating and heat rejection of the solid parts in the engine, derived from the fuel internal energy, further physical tests is required. The necessary analyses would be investigations of how the created energy in the combustion is distributed and absorbed by the engine parts. It would also include measurements of the convection effects of the underhood air flow on the engine block and radiation from the engine solid parts.

The component data in our Flowmaster model needs to be updated to reflect the behavior of the constant changing of the physical components. Primarily an evaluation of components such as the water pump characteristics and pressure losses in the coolant circuit is needed. An investigation of the emitted heat from the condenser and engine heat rejection maps which is up to date is also necessary.

Table of reference

[1] Cengel, T.A. and Boles, M.A. (1998) *Thermodynamics: an engineering approach.* (3. ed.) Hightstown: McGraw-Hill. ISBN 0-07-115247-4

[2] *Flowmaster On-line reference Help.* Flowmaster International Ltd

[3] Incropera, F.P. and DeWitt, D.P. (2002) *Fundamentals of heat and mass transfer.* (5. ed.) New York: John Wikey & sons. ISBN 0-471-38650-2

[4] Moffat, J. (2002) Coupling of 1-D and 3-D CFD-models to predict transient hydraulics in a engine cooling circuit. *SAE 2002 World Congress*, Detroit, March 4-7 2002. ISSN 0148-7191

[5] Nilsson, C. (2004) *Coolant fan control EUCD.* Gothenburg, Volvo Car Corporation

[6] Stone, R. (1999) *Introduction to internal combustion engines.* (3. ed.) New York: Palgrave. ISBN 0-333-74013-0

[7] White, F.M. (1999) *Fluid mechanics.* (4. ed.) Hightstown: McGraw-Hill. ISBN 0-07-116848-6

[8] Williams, J. and Karanth, D. (2002) Cooling inlet aerodynamic performance and system resistance. *SAE 2002 World Congress*, Detroit, March 4-7 2002. ISSN 0148-7191

[9] Hermansson, J (2005) *CFD-report*, Gothenburg, Volvo Car Corporation

Appendix

Appendix A – Flowmaster model of the cooling system for the SI6 NA engine with manual gearbox

Figure A.1 illustrates the complete Flowmaster network for a SI6 NA engine with manual gearbox. The system is described in detail in Appendix C to F.

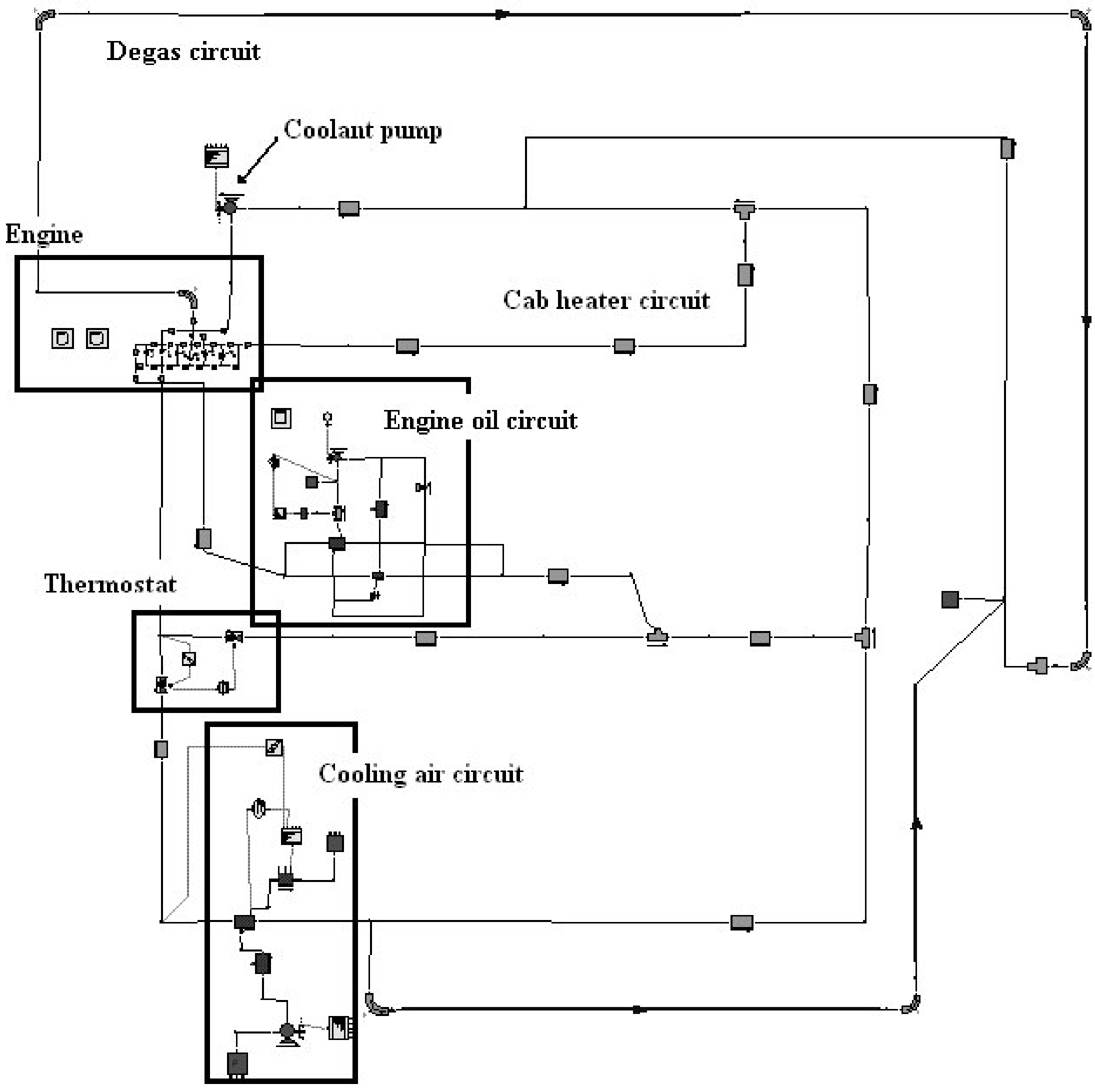

Figure A.1 Coolant circuit for the SI6 NA engine with manual gearbox modelled in Flowmaster

Appendix B – Flowmaster model of the cooling system for the SI6 Turbo engine with manual gearbox

Figure B.1 illustrates the complete Flowmaster network for a SI6 Turbo engine with manual gearbox. The system is described in detail in Appendix C to F.

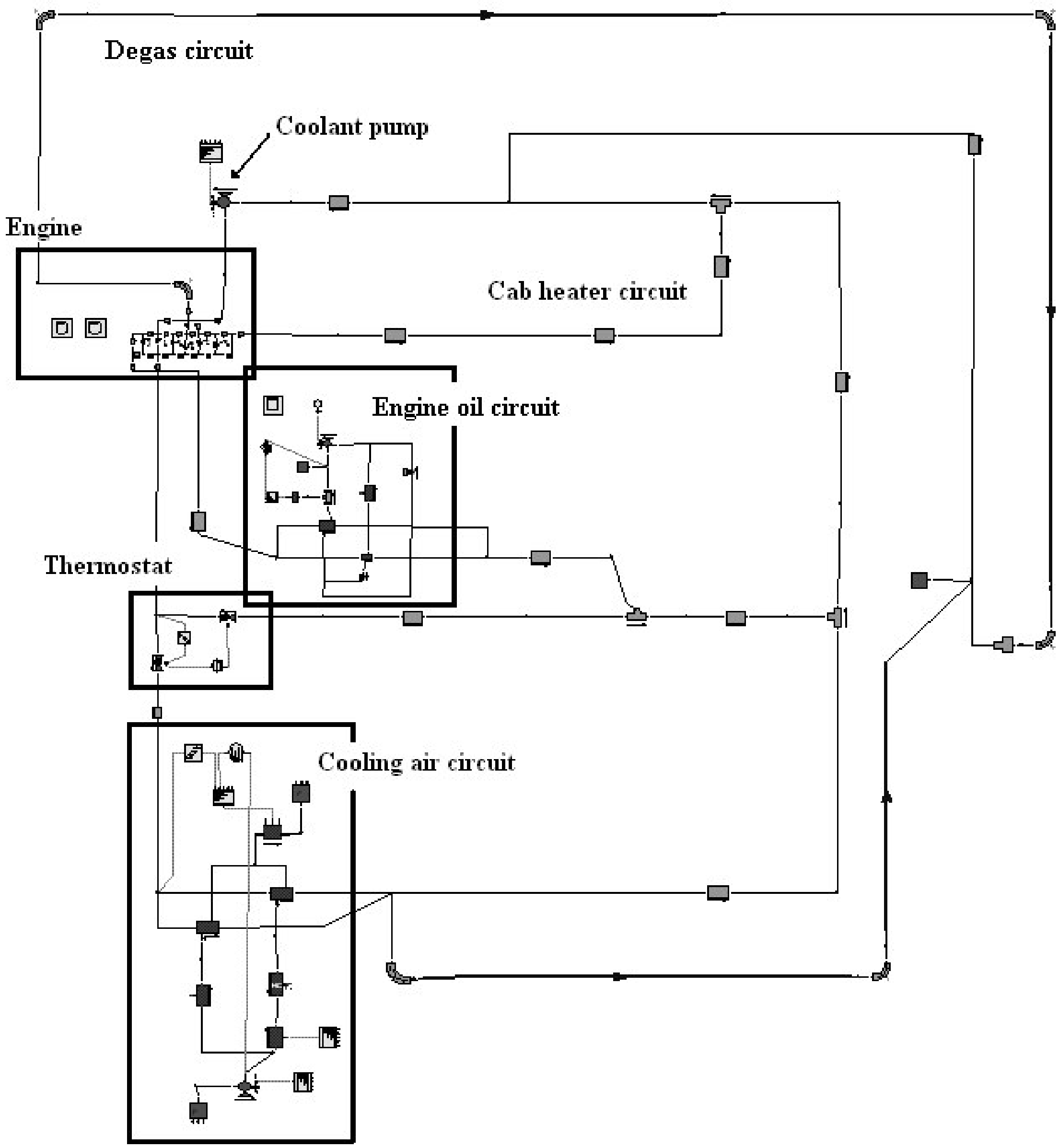

Figure B.1 Coolant circuit for the SI6 T engine with manual gearbox modelled in Flowmaster

Appendix C – SI6 engine

The engine is modeled in Flowmaster as a network of *discrete loss* components regulating the coolant flow through the engine. This structure is applied on both the SI6 NA and Turbo charged engine configuration. To be able to simulate an engine during transient conditions *controllers* is used to control the components. The *controllers* gather information (engine rpm and engine load) from the *engine* and *vehicle components* through out a script (Figure C.1). These parameters is translated to required emitted energy to the coolant at the particular driving condition via a 3D-engine map in the engine component, see Figure C.2. Energy is transferred to the coolant by the h*eat source* and t*hermal bridge* components. The difference between the two engine configurations is the amount of heat transferred to the coolant.

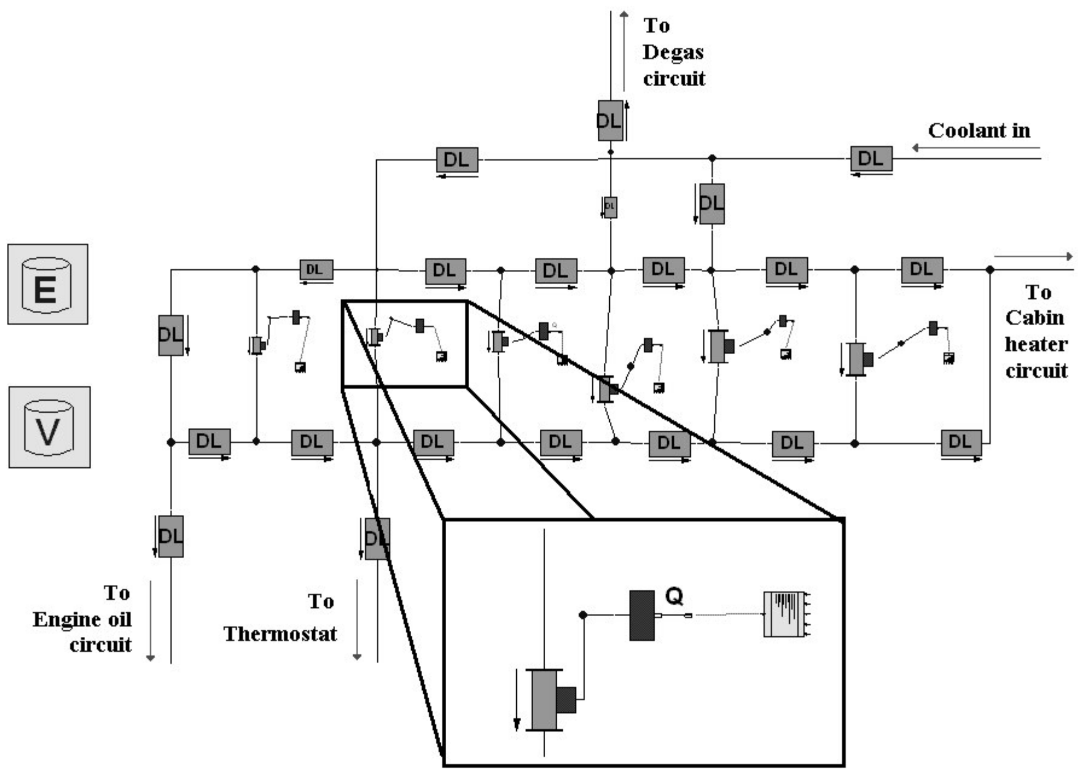

Figure C.1 SI6 engine

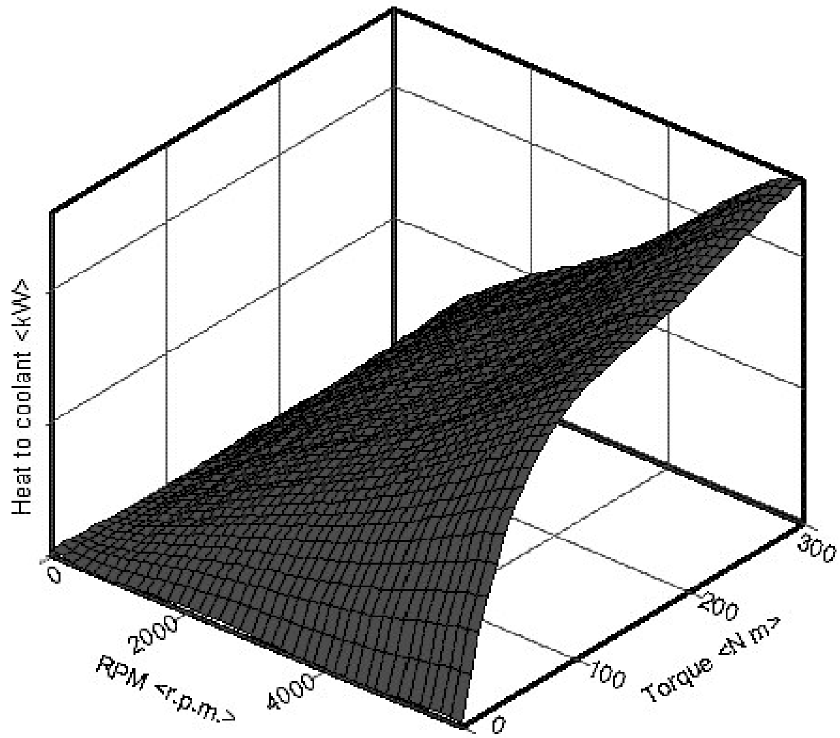

Figure C.2 Generated heat from engine (per cylinder) to coolant for SI6 NA

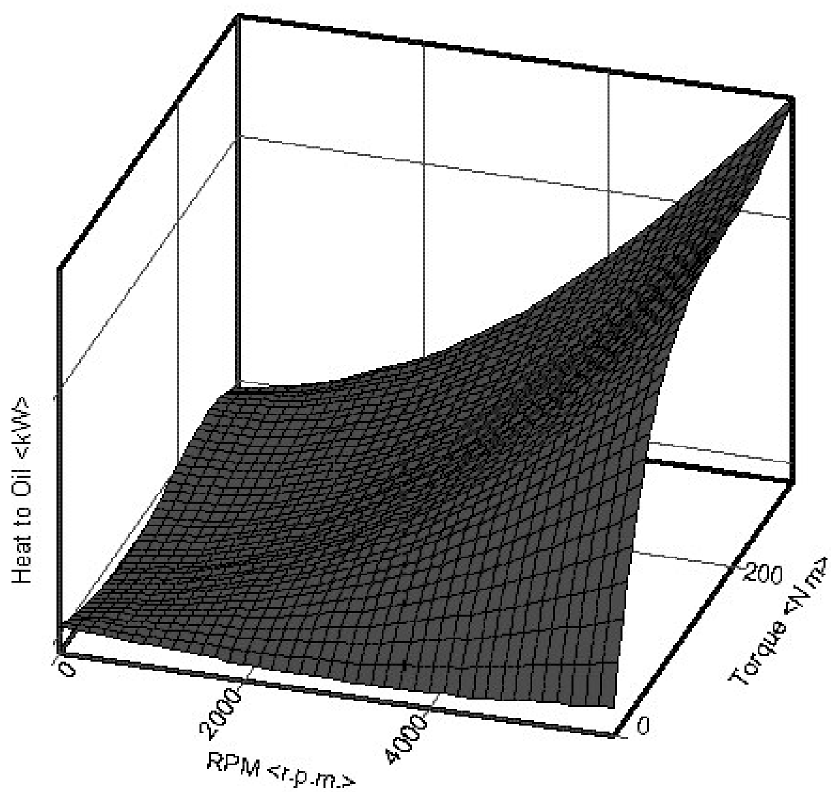

Figure C.3 Generated heat from engine to engine oil for SI6 NA

These 3D-engine maps were obtained from an engine test made previous to our thesis.

Appendix D – Engine Oil circuit

The Flowmaster model of the engine oil circuit contains a *radial flow pump* at constant rpm producing the correct oil flow (Figure D.2), two *thermal heat exchangers* where heat is transferred from the oil to the coolant and a *thermal bridge* component supplying the oil with energy in the same manner as the cylinders. A *controller* component gathers information from the upper node and the e*ngine component* to control the generated heat flux in the *thermal bridge.*

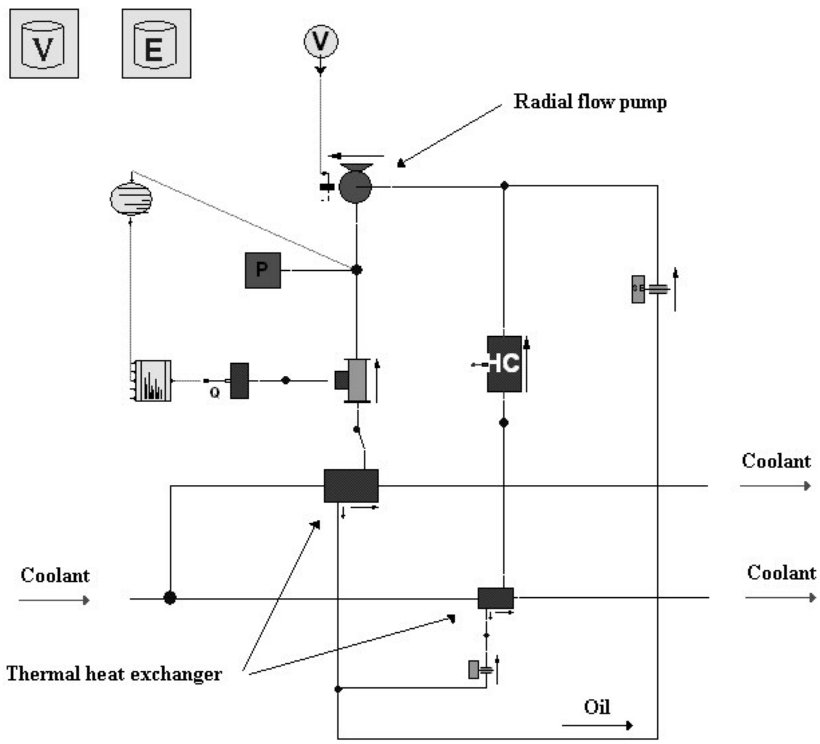

Figure D.1 Engine oil circuit

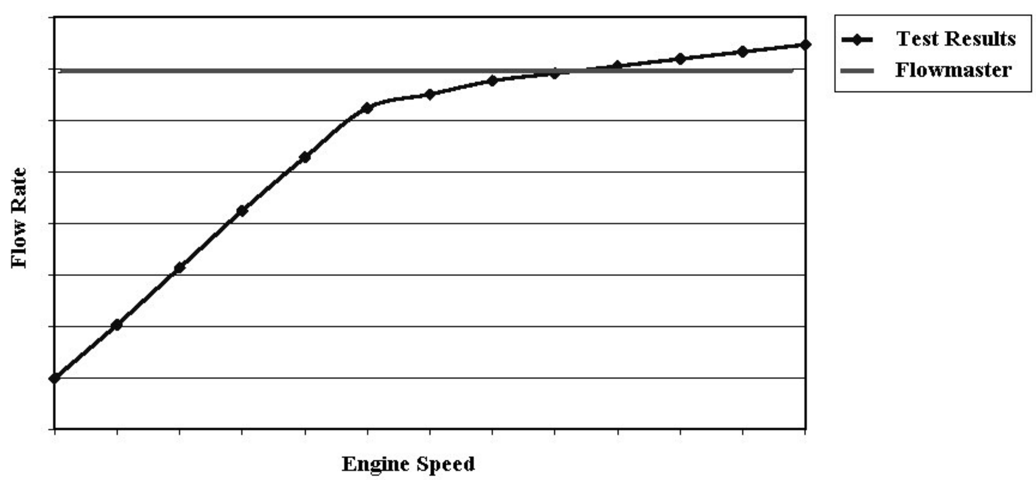

Figure D.2 Oil flow through engine oil pump

Appendix E – Thermostat

The thermostat controls the coolant flow through the radiator circuit and the by-pass circuit. To simulate the thermostat in Flowmaster the component setup illustrated in Figure E.1 was used.

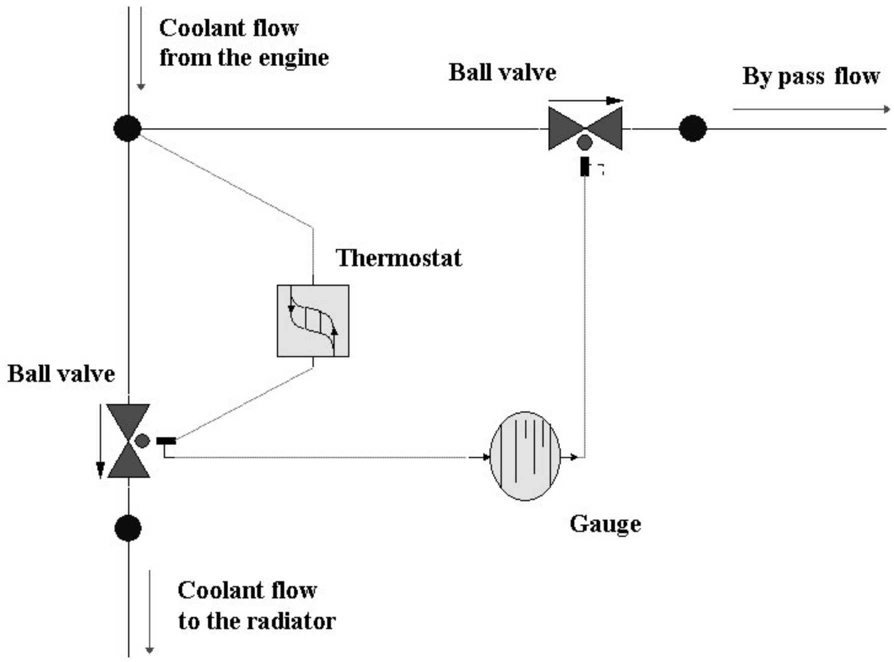

Figure E.1 Thermostat

The *thermostat* regulates the coolant flow to the radiator with respect to the temperature of the coolant from the engine. The *gauge* orders the *ball valve* for the by-pass flow to open inverse to the *ball valve* controlled by the *thermostat*. The *thermostat* contains two hysteresis curves to model the different behavior at increasing and decreasing temperature respectively. The hysteresis curves are used to prevent rapid changes in coolant temperature.

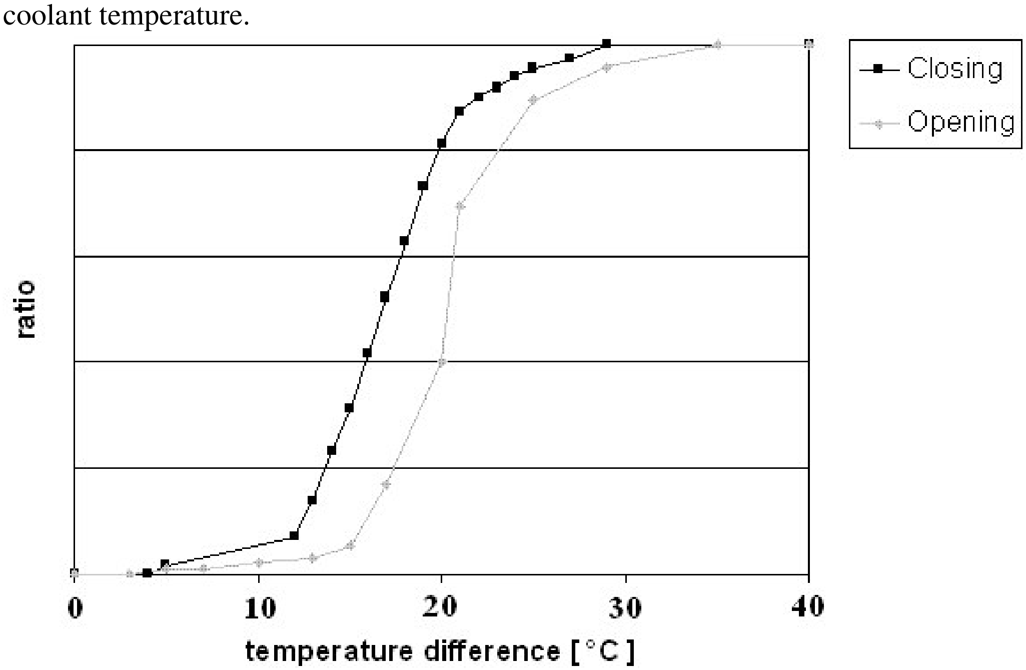

Figure E.2 Thermostat hysteresis curves

Appendix F – Air cooling circuit for the SI6 engine with manual gearbox

For the SI6 engine with manual gearbox there is two different configurations of the air cooling circuit depending on whether the engine is naturally aspirated or Turbo charged.

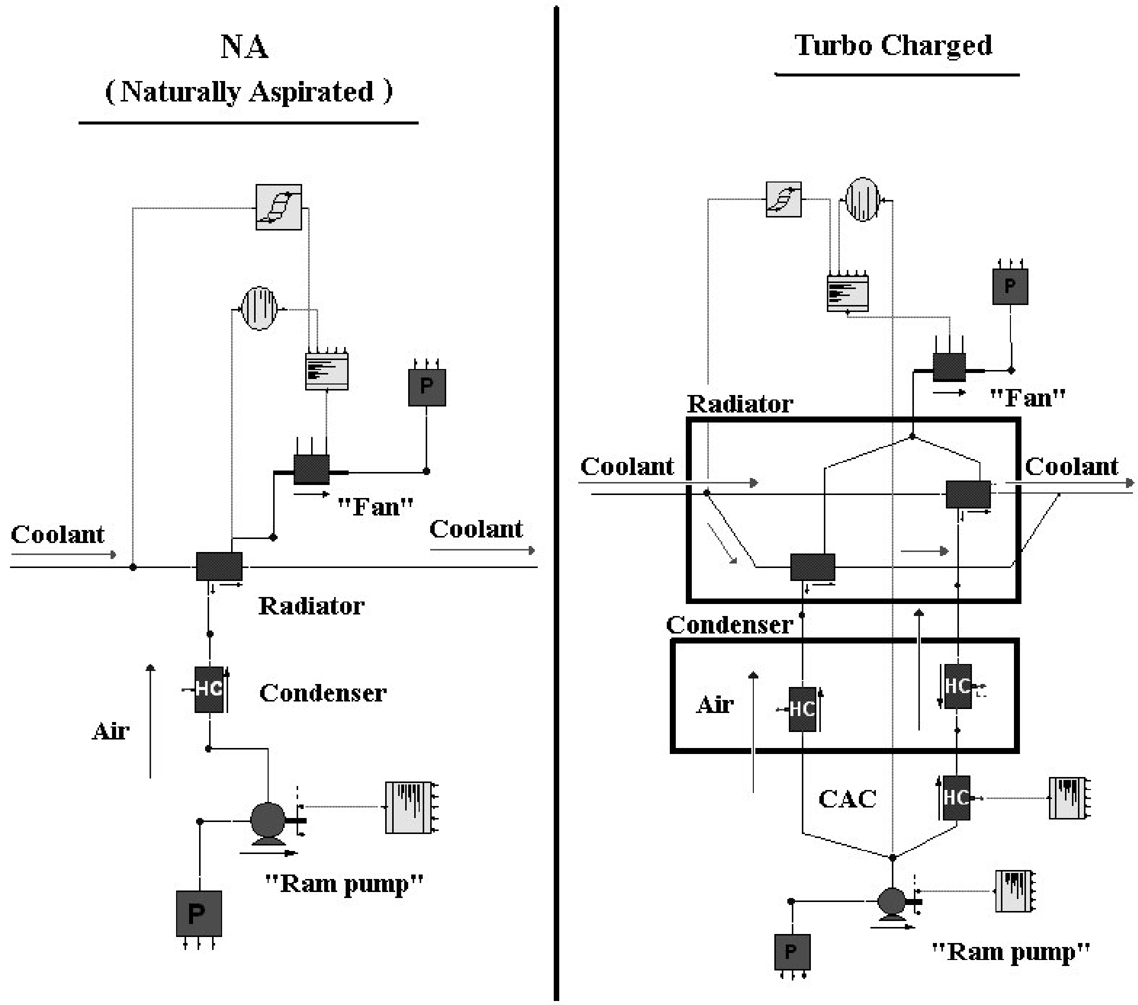

Figure F.1 Air cooling circuit

For both engines the air flow enters the circuit through the grille represented by the *ram pump* in Flowmaster. Then, for the turbo charged engine the air flow is divided into two paths. The right path in Figure F.1 represent the air going through the *CAC*, and the left path the air going directly to the *condenser*. The *radiator* is also divided into two components representing the part behind the CAC and the rest of the radiator respectively. In the radiator (a t*hermal heat exchanger*) heat is transferred from the coolant to the air.

Appendix G – Flowmaster model of the cooling system for the SI6 NA engine with automatic gearbox

The SI6 engine with automatic gearbox includes a transmission oil circuit where the transmission oil is cooled by coolant, diverted from the radiator. To be able to simulate this the radiator have to be divided into an additional part in the Flowmaster model. Figure G.1 illustrates the complete Flowmaster network for a SI6 NA engine with automatic gearbox. The system is described in detail in Appendix I and J.

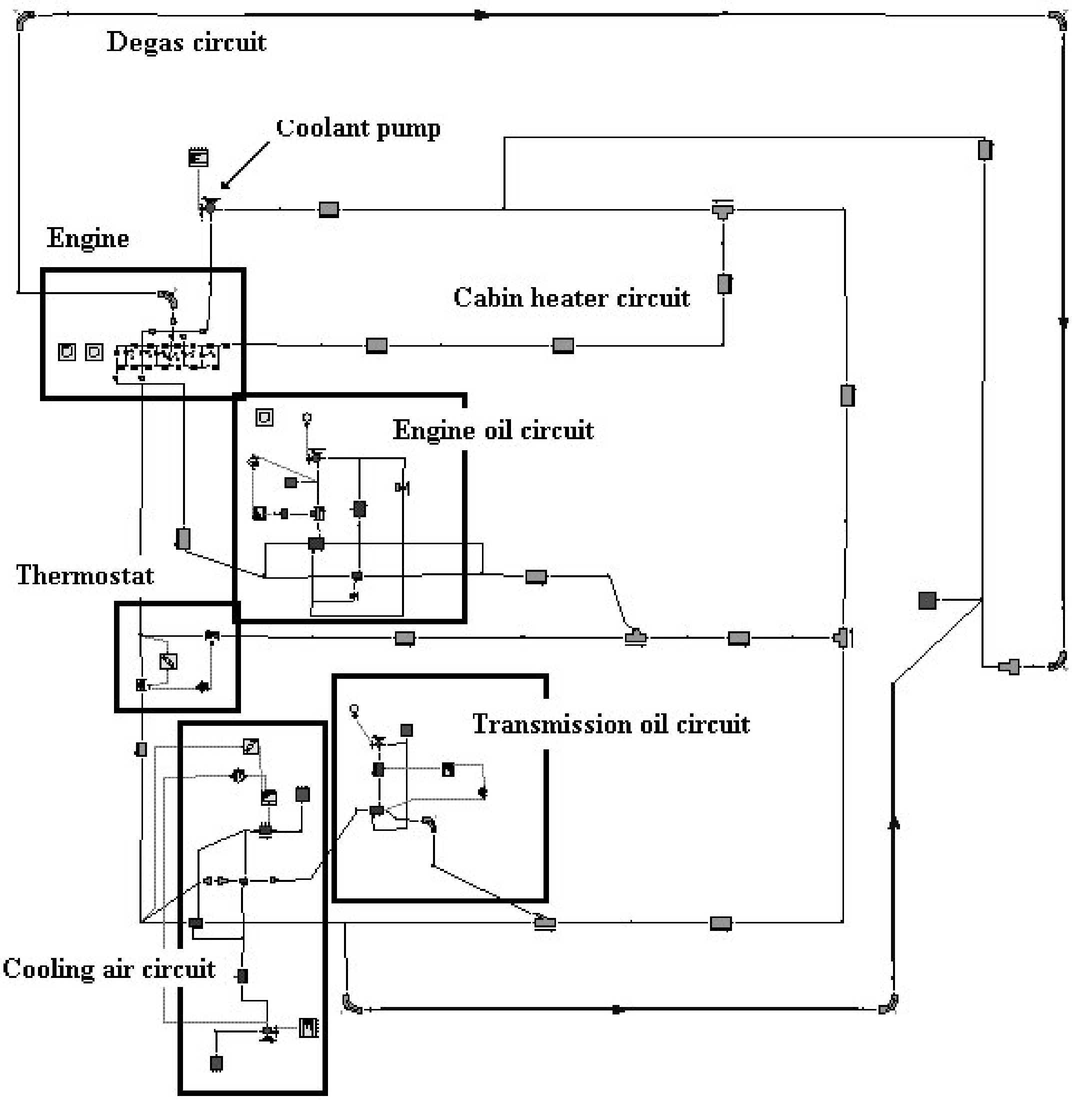

Figure G.1 SI6 NA engine with automatic gearbox

Appendix H – Flowmaster model of the cooling system for the SI6 Turbo engine with automatic gearbox

Figure H.1 illustrates the complete Flowmaster network for a SI6 T engine with automatic gearbox. The system is described in detail in Appendix I and J.

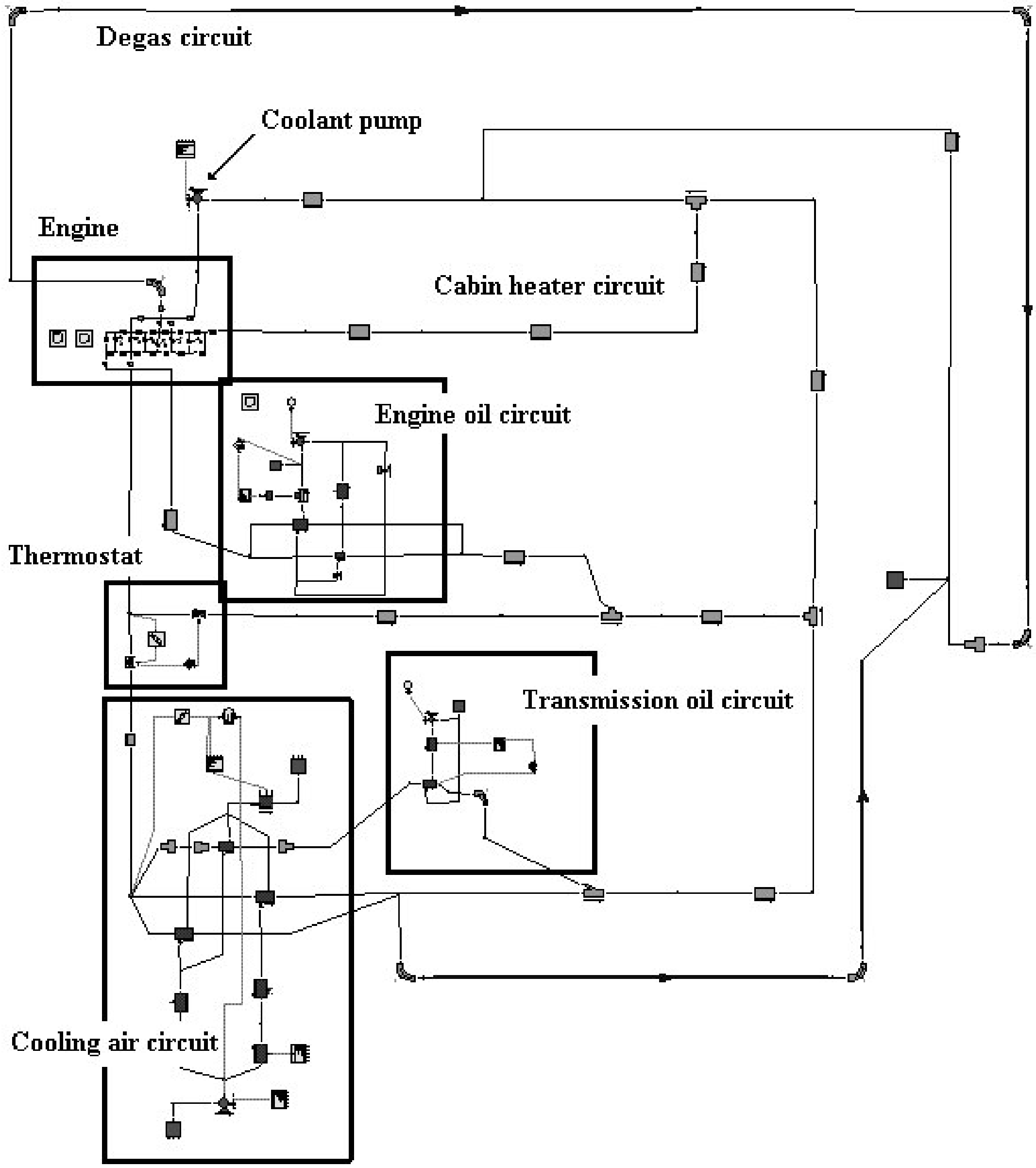

Figure H.1 SI6 Turbo engine with automatic gearbox

Appendix I – Transmission Oil Circuit

To simulate the created frictional heat in the transmission oil circuit, energy is transferred to the transmission oil via a *heater-cooler (heat exchanger)* component. The amount of transferred energy is set by the *controller*, and the *gauge* measures the coolant flow through the *thermal heat exchanger* component. If there is no coolant flow through the heat exchanger, no energy will be transferred to the transmission oil. This is a device to assure that the Flowmaster model converge properly. The *thermal* component represents the transmission oil cooler, WTOC, where heat is transferred from the transmission oil to the coolant.

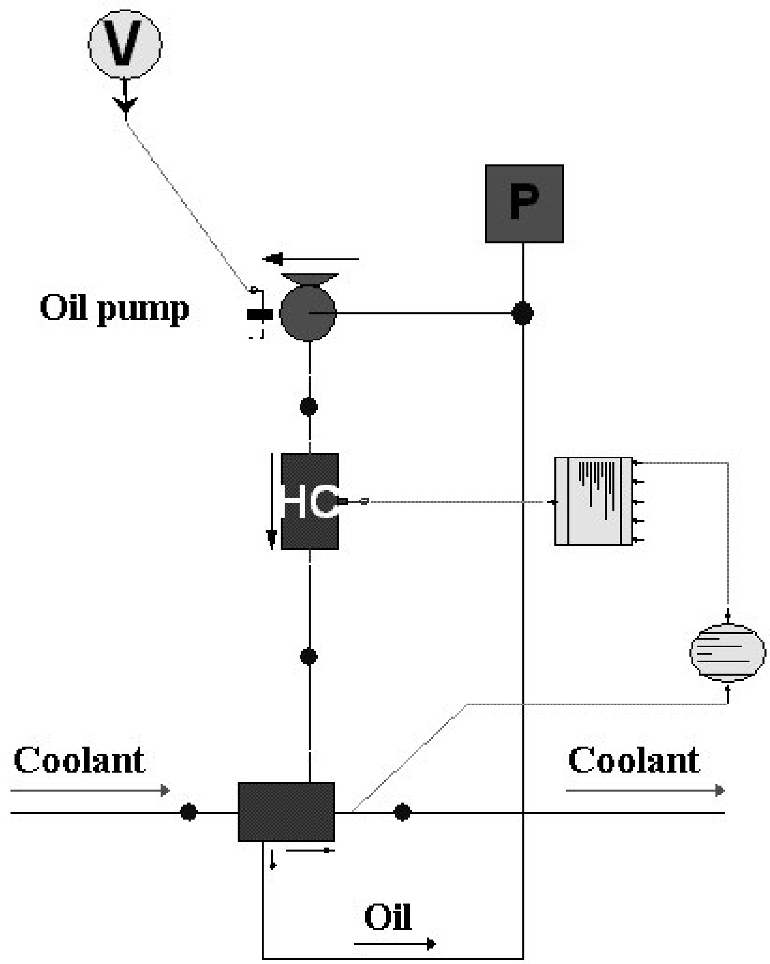

Figure I.1 Transmission oil circuit

Appendix J – Air cooling circuit for the SI6 engine with automatic gearbox

The two different air cooling circuits for the SI6 engine with automatic gearbox are shown in Figure J.1. The Flowmaster model for the SI6 engine with an automatic gearbox has an additional air path for the cooling air, which means that the radiator has to be divided into one more section. This part is represented by the upper heat exchanger which is called the *subcooler*. The coolant flowing via the *subcooler* is diverted to the WTOC for cooling of the transmission oil.

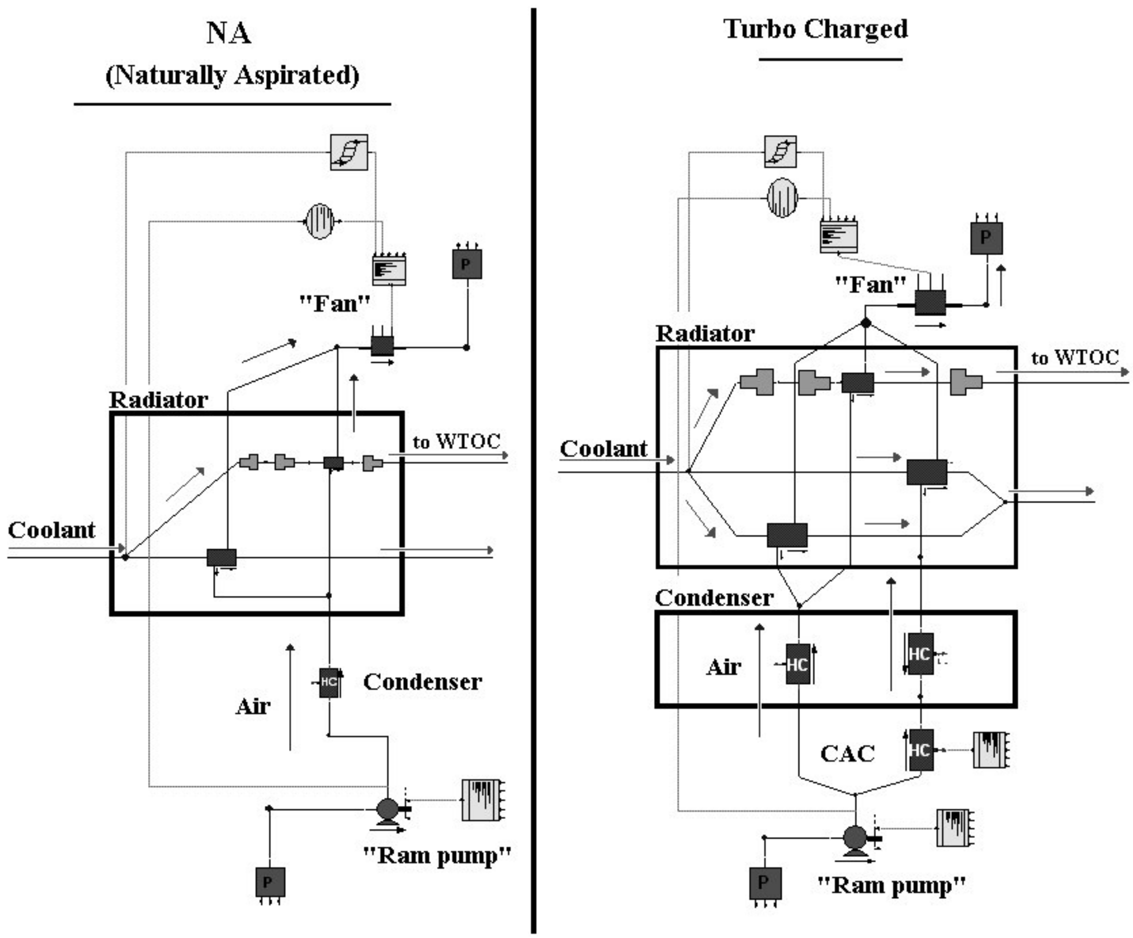

Figure J.1 Air cooling circuit for the SI6 engine with automatic gearbox

Appendix K – Component data

Fan

Figure K.1 and Figure K.2 illustrates the fan curves for SI6 NA and Turbo respectively.

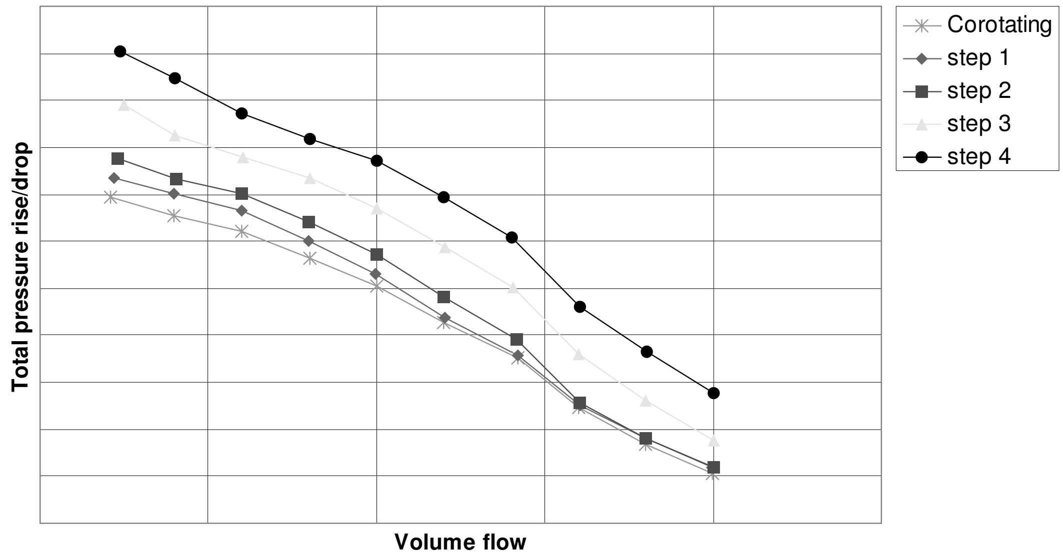

Figure K.1 Fan curves for single fan

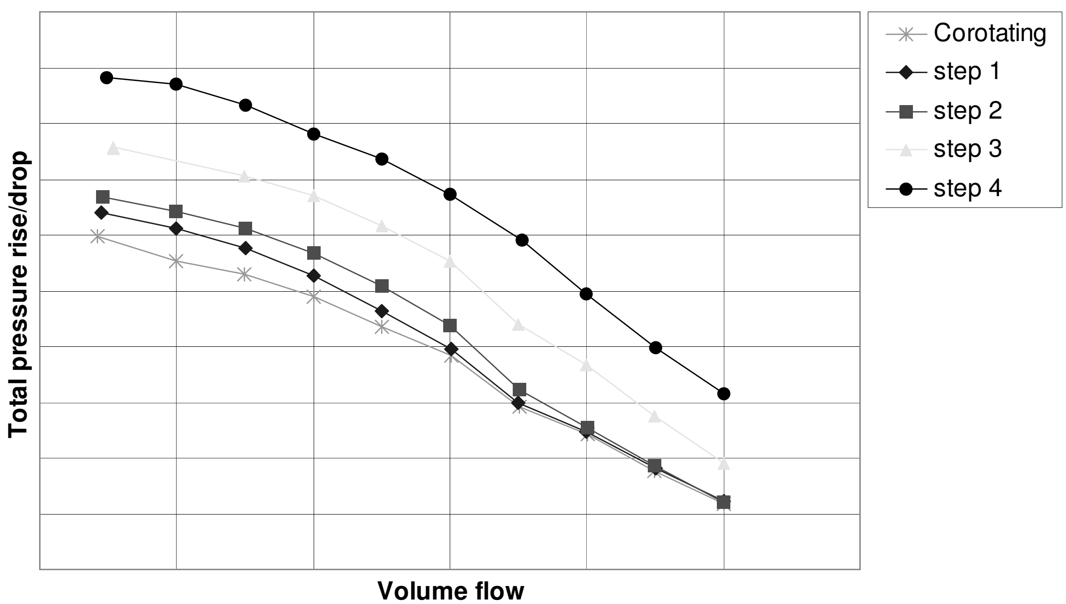

Figure K.2 Fan curves for double fan