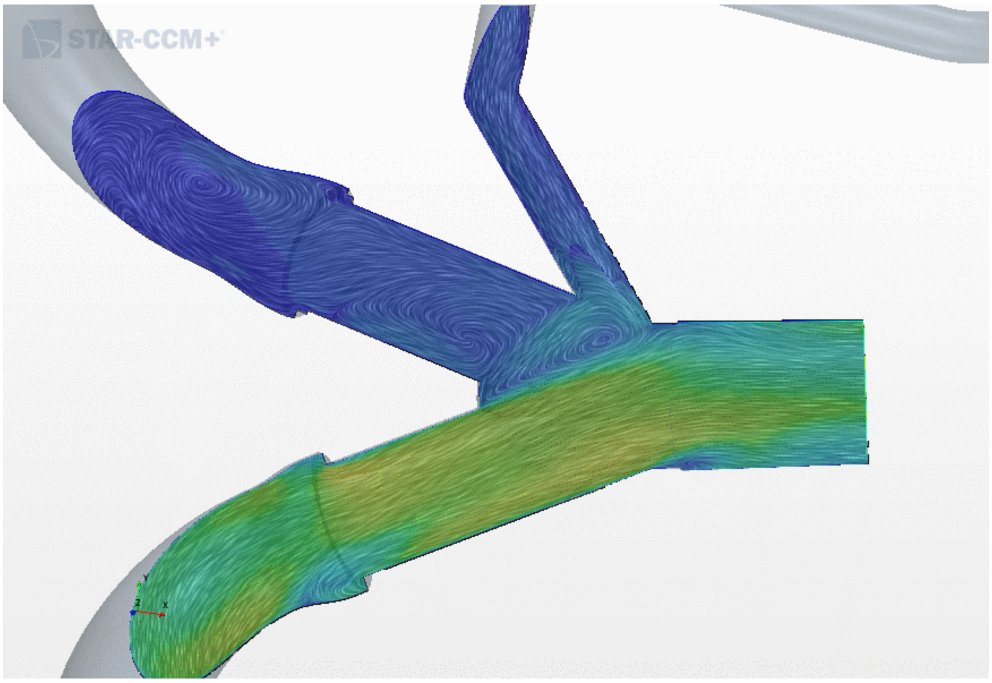
CHALMERS



Implementation and validation of 1D-3D CFD co-simulation for complete cooling system

Master's thesis in Applied Mechanics

KONSTANTINOS KONSTANTINIDIS

Department of Mechanics and Maritime Sciences
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg, Sweden 2020

MASTER'S THESIS IN APPLIED MECHANICS

Implementation and validation of 1D-3D CFD co-simulation for complete cooling system

KONSTANTINOS KONSTANTINIDIS

Department of Mechanics and Maritime Sciences
Division of Fluid dynamics
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2020

Implementation and validation of 1D-3D CFD co-simulation for complete cooling system
KONSTANTINOS KONSTANTINIDIS

Master's thesis 2020:80
Department of Mechanics and Maritime Sciences
Division of Fluid dynamics
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: +46 (0)31-772 1000

Cover:
From 3D CFD simulation: A vector plot of the velocity on a middle plain of a flow splitter.

Chalmers Reproservice
Göteborg, Sweden 2020

Implementation and validation of 1D-3D CFD co-simulation for complete cooling system
Master's thesis in Applied Mechanics
KONSTANTINOS KONSTANTINIDIS
Department of Mechanics and Maritime Sciences
Division of Fluid dynamics
Chalmers University of Technology

Abstract

This thesis deals with the development and verification of a method that introduces coupling between one- and three- dimensional (1D and 3D) numerical simulations of a complete cooling system of a passenger car, as well as with the advantages of using such a platform. Such method is different from the common practise in the car industry, in which simplified models are typically used and they consider each component of the system as 1D. These models are typically based on emperical correlations. However, the 3D models solve for the differential equations that govern the fluid dynamics of a single-phase coolant and offer more detail in the analysis of a system. They are used for a part of the cooling system that the 1D models fail to capture the behaviour of the fluid and the physics. The rest of the cooling circuit will stay 1D, creating a need to device a method to couple 1D and 3D Computational Fluid Dynamics (CFD) software. The project was performed in the Cooling system development group at Volvo Cars. First, an analysis of the details of the cooling system and its modelling theory with 1D CFD models are introduced. Moreover, turbulence and its treatment in 3D CFD are described, while the theory behind coupling two CFD software is thoroughly established. The method for coupling has been developed with GT-SUITE for the 1D and STAR-CCM+ for the 3D. The inputs to the system, for example wall boundaries or operating points, are constant in time. A fully functional co-simulation platform is tested with different configurations. In each of them, a different component of the cooling system is simulated in 3D while the rest circuit runs in 1D. The results show stability in the co-simulation of all the configurations at and near the point that the coupling of the softwares occurs. At next, a discussion is made about the possible limitations of 1D modeling for these cases, with the conclusion of the advantages that were observed from the co-simulation in contrast to the extra computational cost it emerged. Lastly, there is an introduction of cases where cavity or boiling occurs in the cooling system, that require multiphase flow simulations, but the limitations in coupling the softwares didn't allow further simulations. An investigation could be further performed in the future, following the assumption that the software will upgrade their co-simulation capabilities

Keywords: Co-simulation, cooling system, coupling, GT-SUITE, 1D CFD, STAR-CCM+, 3D CFD

Preface

This master's thesis was part of a project by the System development Group (97494) with focus on the vehicles' cooling system at Volvo Cars Corporation, Gothenburg, Sweden. The project focused on developing and testing a simulations' platform, as well creating a manual for further use within the department. It was performed by two students from different universities, Amirali Kerachian, an MSc Vehicle engineering student in Kungliga Tekniska Högskolan (KTH) and the author of the thesis from Chalmers tekniska högskola. Two different thesis were based on this project. It was supervised in its entirety by CAE engineer at 97494, Juan Antonio Moreno Gomez. The examiner of the thesis was Professor Srdjan Sasic from the division of Fluid dynamics.

Acknowledgements

I would like to equally acknowledge and praise the efforts, knowledge and technological expertise of my project partner, fellow student, Amirali Kerachian and my supervisor Juan Antonio Moreno Gomez. They helped to create a concrete plan, set goals and eventually achieve the objective of this project. Moreover, the thesis was performed with remote work, therefore I would like to thank them for not letting this affect our collaboration and keeping the spirit high.

I would also like to thank the manager in the 97494 group Linda Bodforss for giving me the opportunity to work both as a summer worker and as a master thesis student to her department in Volvo Cars. It was a great experience for me and I would like to wish for the department and for her personally the best for the future.

Furthermore, I would like to express my special thanks to my examiner, Srdjan Sasic, for our co-operation and mutual understanding of the tight schedule for both and of the problems which this thesis occasionally faced throughout the spring semester.

Konstantinos Konstantinidis
Gothenburg, June 2020

Contents

List of Figures

Nomenclature

ΔP	Pressure raise or drop, absolute difference of pressure at inlet and outlet	Pa
$\dot{q}$	Heat source per mass unit	$\mathrm{J/kg \cdot s}$
ϵ	Turbulent dissipation rate	$\mathrm{J/kg \cdot s}$
$\mathbf{v}$	Velocity	m/s
ω	Specific dissipation rate	1/s
ρ	Density	$\mathrm{kg/m^3}$
$\underline{\underline{\tau}}$	Shear stress tensor	Pa
e_o	Specific total internal energy	J/kg
g	Gravitional acceleration	$\mathrm{m/s^2}$
H	Delivery head	m
h_o	Specific total enthalpy	J/kg
k	Kinetic energy of turbulent scales	J/kg
k-ω SST	k-ω Shear stress transport model	
P	Pressure	Pa
Q	Volumetric flow	$\mathrm{m^3/s}$
T	Temperature	K
y^+	Non-dimensional normal distance from wall	
Re	Non-dimensional Reynolds number	
1D	One-dimensional	
3D	Three-dimensional	
CFD	Computational fluid dynamics	
EGR	Exhaust gas recirculation system	
EOC	Engine oil cooling module	
FVM	Finite volume method	
GUI	Graphic user interface	
HVAC	Heating, ventilation and air-conditioning system	
ICE	Internal combustion engine	
RANS	Reynolds Averaged Navier Stokes method	
TOC	Transmission oil cooling module	
VOF	Volume of fluids	

Part I
Introduction, Theory and Methodology

1 Introduction

1.1 Background

The water cooling systems that are used in vehicles are designed with a closed circuit in which the coolant recirculates. The liquid will absorb the heat from the engine and several other components and dispose it to the environment in the component that interacts with it, the radiator. To analyze the behaviour of the system, validate its functionality or optimize the design, a robust simulation platform is required. Some of the desired features of such a platform are: i) accuracy of the parameters of the system within an acceptable region, ii) ability to model all or most of relevant physics and geometries, iii) to offer a user-friendly interface to analyze and present results and iv) to minimize computational time.

Computational fluid dynamics(CFD) is the field that researches the flow behaviour by simulations. The finite volume method (FVM) in two and three dimensions is the most often used methods to predict the flow, but several others also exist and are used either in the academia or the industry. In the development of the cooling system specifically, it is widely used by the automotive industry an 1D CFD solver that contains models to adjust the equations for geometry changes, moving components (such as pumps or valves), turbulence effects and others. These adjustments have emerged from experimental data. The user is required to tune the models for certain sub-components by providing his experimental data. After tuning, the 1D platform contains almost all the desired features for cooling simulations, scoring very well on computational time but lacking on the ability to model all physical phenomena observed in cooling systems.

More specifically, simulations that include multiphase phenomena are available but with immense simplifications in the modeling of these phenomena. Models that have been developed for 3D fluid dynamics simulations are more reliable. To perform a full 3D simulation in the closed circuit of the cooling system will require very high computational resources, while flexibility for modifications is considered limited. The concept that attracts interest is to combine the two type of simulations and to focus the 3D only to locations that higher detail is required.

Such type of co-simulations have been investigated in the past. The simulations on the engine, the intake manifold and the exhaust gas recirculation (EGR) at the regions of inflow, combustion and outflow of gas in a combustion engine, require to be simulated with 3D CFD. However, the flow in the piping between the 3D parts of the system or other simplified geometries behaves one-dimensional and therefore is able to be accurately modelled with 1D simulations. As a result coupling of the two simulations has been used in the automotive industry, improving computational cost and flexibility of the system. [7]

1.2 Problem description

The department responsible to model and perform simulations over the complete cooling system of the vehicle in Volvo Cars is using 1D software, that is able to provide the heat rejection in various sub-systems. To investigate some irregular phenomena that can be observed experimentally, they can only perform a 3D simulation on a restricted part of the system that must be continuous. There are limitations occurring for these cases, since certain phenomena in the flow can be created in different locations and cause problems further downstream. Moreover, the transient behaviour that is sometimes researched for a cooling system, cannot be taken into account from the boundary conditions of the 3D.

The project focuses on creating a platform for running simultaneously an 1D flow software, modeling the majority of the cooling system, and a 3D CFD software, which can solve for higher, detailed, information in its domain. The project is constituted of the following parts. At first, there is a small analysis on the cooling system and the benefits of partially running 3D, some theory about possible phenomena that can be observed, as well as their modeling methods. The main part is dedicated to create the method that establishes a coupling platform and to further validate it with a verification process. The verification is performed by testing the co-simulation platform, comparing with single 1D data from models that have been verified with experimental data. This will permit the use of the co-simulation to capture the 3D details in different sub-components that are selected and simulated. Initially, focus of the coupling simulations is given on simple geometric components and physics, with testing it over a variety of cases and co-simulation settings that can occur in the system, such as different pump speeds, valve positions and others. In the last stage of the project, a section that may contain several sub-components of cooling is simulated with also including a transient change in a valve of the 1D system.

The obective of this project is to establish and verify the method to perform co-simulations in cooling systems, in order to permit the use of higher fidelity models on simulating the behaviour of the cooling systems. It will allow simulations under different loads in steady and transient cases that may investigate phenomena, that are unable to be captured correctly by 1D simulations.

1.3 Limitations

When limitations are mentioned while performing CFD simulations, they can be analyzed through two aspects, the modeling accuracy and the computational resources. For this project the validity of the existing 1D model has been compared and validated with experimental results for the majority of the cases. Moreover GT-SUITE incorporates the possibility of preserving a mixture of different species, fluids or even phases. As a result, for this project it will be considered that only from the 3D CFD software STAR-CCM+ limitations are identified. Also, there is the limitation that there are not experimental data available for the simulations that are planned. As a result, the results of the co-simulation won't be able to be verified if they perform more accurately in comparison to the 1D model.

The models for turbulence modeling that are offered in STAR-CCM+ are the classical turbulence treatment models plus some more advanced that will be not investigated from this thesis. They could have focused on identifying small scale phenomena but that would increase the number of cells and the computational cost enormously. To capture multiphase phenomena the software offers the following models: Lagrangian Particle Tracking (LPT), eulerian sub-methods and Volume Of Fluids (VOF). In simulations for cooling system, possible multiphase phenomena are the existence of gas (air or vapour) in liquid water. Vapour creation can be observed by cases that include nucleate boiling or cavitation. The former is offered for both a single-phase liquid solver and in an Euler-Euler segregated solver, while the latter is offered by VOF implementation. Lastly, air existence and the phenomenon of sloshing (fluid movement within a tank) in the gas expansion system could be modeled by VOF or, if required, other Euler methods. Some of these cases would have been tested in the current thesis, but the 3D software did not offer the explicit coupling of the two softwares in combination with its multiphase models.

Further limitation of STAR-CCM+ is the computational cost since it can't be discretized for the complete domain of the cooling system. If there will be a breakthrough in computational performance, then 1D systems may be considered obsolete. Unfortunately, the biggest restriction for this project comes for the automated coupling abilities of STAR-CCM+. It offers coupling only for single-phase liquid solver and not for multiphase solvers. The software's developing team should add the feature of sending and receiving boundary conditions that can include mass fractions of the different phases. Therefore the project focused on co-simulating with a single-phase coolant and investigate the effect of complex geometry in implementing 3D CFD.

2 Theory

2.1 The cooling system

2.1.1 Introduction of the cooling system in vehicles

The cooling of the driving unit and other components of a vehicle is most usually designed with a forced recirculation of water. The fluid passes through specially designed cavities surrounded by metal surfaces to maximize heat transfer from the components to the coolant. To connect the cooling circuit and create a closed system hoses, connectors and flow splitters are used with circular cross-section. The heat that the liquid absorbs is released in the environment at the radiator, an air to water heat exchanger that will be explained further in following chapter. In order to avoid freezing in very low temperature conditions, there is addition of ethylene glycol at a fraction of 30 to 50 % of the total mixture. It has the ability to reduce the freezing point of the mixture as low as -35 ^{o}C. The mass flow of the coolant can be forced by a pump that can be either belt-driven or electrical. An example of a cooling system representation is depicted in Figure 2.1.

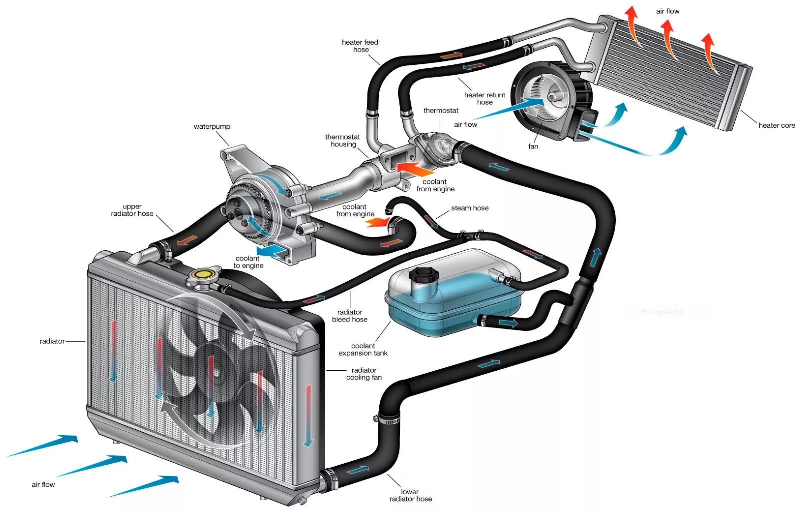


Figure 2.1: *Example of a cooling system in a passenger car. Source engineeringdiscoveries.com [3]*

Some other significant applications of the cooling system is to keep the engine in a temperature that increases its overall efficiency, to provide the cooling of the oil in order to retain its lubricating properties, as well as the heating of the cabin through a heat-exchanger which is part of the climate system, often mentioned as 'Heating, ventilation and air-conditioning' or HVAC. Furthermore it is used to cool the part of the exhaust gases that will be recirculated to the engine inlet and, moreover, to cool (or else mentioned in literature as "charge") the inlet air at the exit of the compressor through the water intercooler. Lately the cooling system is also used to keep a relatively constant temperature in the batteries and the rest of electrical and electronic systems of electrically driven vehicles, since operating at at a certain span of temperatures increases the efficiency and life-span of these components.

2.1.2 Important sub-systems

Engine block cooling - Waterjacket

The combustion heat that is absorbed by the cylinder walls inside the engine can raise high temperatures and therefore the engine requires cooling. Otherwise the metal in the engine block will expand and deform with catastrophic consequences for the engine. The engine block is designed so oil and coolant will flow in channels between metal parts of the engine. The channels for the coolant that surround the cylinders comprise the waterjacket.[9] Heat transfer efficiency and pressure drop of the coolant inside the waterjacket are very important parameters for cooling in internal combustion engines (ICE). Several 3D CFD simulations are performed to design and optimize the geometry of the waterjacket. The heat treansfer in the walls is simulated with Finite Element Method (FEM). Several investigations for coupling 1D and 3D cooling simulations. The combustion heat that is absorbed by the cylinder walls inside the engine can raise high temperatures and therefore the engine requires cooling. Otherwise the metal in the engine block will expand and deform with catastrophic consequences for the engine. The engine block is designed so oil and coolant will flow in channels between metal parts of the engine. The channels for the coolant that surround the cylinders comprise the waterjacket. Heat transfer efficiency and pressure drop of the coolant inside the waterjacket are very important parameters for cooling in internal combustion engines. Several 3D CFD simulations are performed to design and optimize the geometry of the waterjacket. The heat treansfer in the walls is simulated with Finite Element Method (FEM).

Investigations for coupling 1D and 3D cooling simulations have been introduced where the waterjacket was the part modeled in 3D[11]. This thesis was limited in performing 3D simulations in the waterjacket since it belongs to a different group inside the RnD of Volvo Cars. This thesis was limited in performing 3D simulations in the waterjacket since it belongs to a different group inside the RnD of Volvo Cars.

Thermostat

The thermostat controls the coolant flow through the pipes and stabilizes the temperature of the engine to its efficient temperature range. It accomplishes that by returning the coolant to the engine in warm-up cases, by giving full coolant flow in the radiator for maximum cooling or by trying to balance the flows in order to keep constant temperature in the engine. Its use becomes of high importance in engine cooling with mechanical pump, since the mass flow going to the engine is unavailable to control. It effectively controls the temperature of the coolant going into the engine when it decides what percentage of the mass flow will pass through the radiator.

Most common configurations of this type of valve is with a wax element, that senses the temperature of the coolant coming out of the engine and operates a flap to permit the coolant to pass through the radiator.

Water pump

The water pump provides the mass flow demand in the circuit under the desired operating condition. It has several points that need to be satisfied for proper use with the most important being the small energy requirement and the avoidance of cavitation phenomena.

The outlet is called the pressure delivery point of the pump, since it provides the fluid to the circuit with raised pressure. It is computed by the delivery head H parameter with length units, making the measurement independent of the density value.

$$H = \frac{\Delta P}{\rho g} \tag{2.1}$$

The power output of the pump is known. For the mechanical belt-driven pumps the torque and engine speed are sufficient to calculate the power given to the pump shaft, whereas in electronic pumps the voltage and current give the power. Moreover the torque is a function of engine speed in the engine, which means that the rotational speed is sufficient to calculate the power in the pump. Taking into account the mechanical losses for the rotation of the pump it can be computed the effective power output given to the fluid. This power output in incompressible flows it can be measured also as the product of pressure raise ΔP and volumetric flow Q through the pump.

When a pump is modelled in 1D systems, experimental data are used to define the head of the pump. The data are recorded for different pairs of pump rotational speeds and volumetric flow. These look-up tables that come up are imported in the system, that reads the pressure rise at the points calculated before and after the pump and the engine speed, giving the flow rate that is needed to achieve this pressure rise. A graph that shows a hypothetical pump curve is given at Figure 2.2. It is seen that for a constant pump speed, the pressure rise is lower for higher mass flow.

Expansion Tank

The coolant storage tank permits the expansion of the coolant when the coolant cools down without harming the components when its density changes. This way it keeps the majority of the circuit filled with water under all operating conditions, while it captures vapor or air gases that have been trapped in the system. Therefore it is located in the highest point in comparison with the closed cooling system. The inlet to the expansion tank is located in the top and the outlet on the bottom to help capturing the gases.

Engine-Transmission Oil Coolant (EOC-TOC)

Liquid to liquid heat exchangers are used to cool the oil from the coolant in the mechanical parts of the engine and transmission. The liquids flow through cylindrical pipes that are connected with metal sheets that transfer the heat from the walls of the oil to the walls of the coolant. Operating the oil in a certain temperature aids to prolong its life and to sustain the desired lubricating performance.

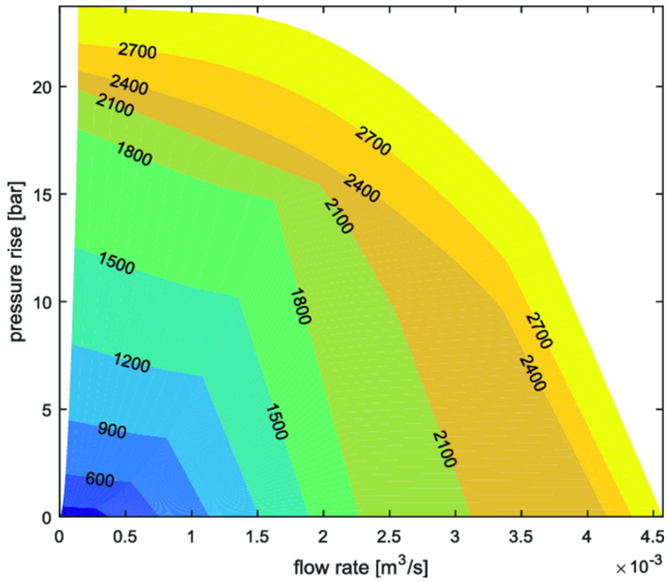


Figure 2.2: *Performance map curves for a pump at different engine speeds.Source publication Marchionni et al.[6]*

Exhaust Gas Recirculation (EGR)

The system that provides the recirculation of a part of the exhaust gases in the intake manifold is known as EGR. It is used mostly in diesel engines, but can also appear in petrol engines. Its function is to reduce emissions by decreasing the fraction of oxygen in the cylinder inlet and by burning unburned carbonhydrates, soot or partially burnt species like CO. Decreasing the percentage of oxygen in a mixture it causes lower combustion temperatures and reduces significantly NO_x emissions. It has proven to be especially effective for the NOx reduction in exhaust gases.

Cooling is needed to reduce the temperature of the exhaust gases before they mix with the air in the inlet. The cooling is performed with a liquid to gas heat exchanger while it requires significantly low temperatures for the incoming coolant.

Heating, Ventilation and Air-Conditioning (HVAC) system in cars

The system on the car that is responsible for providing cold or hot air in the cabin of vehicles is called the climate system. To provide the heating from an ICE to the cabin the vehicle is using the excess heat that is transferred to the coolant of the engine. The coolant is redirected after its interaction with the engine to an air-to water heat-exchanger, also known as heater core. There the air is heated before it would be provided to the cabin.

2.2 Simulations in fluid dynamics

2.2.1 One-dimensional Computational FLuid Dynamics

To perform CFD simulations in 1D, at first a one-dimesional domain needs to be created. The 3D geometry needs to be translated in a single direction model to have the same result as an 1D problem. The equations for conservation of continuity, momentum and energy are solved after creating the 1D domain. In order for these 1D models to simulate the viscosity effect (friction) on the flow and the heat transfer, emperical tables for a variety of geometries, Reynolds number and surface roughness are used. The 1D model tries to estimate a system from a global perspective. When it simulates a complete closed-loop cooling system, it focuses on understanding the mass flow rates, pressures and temperatures that may occur in the different parts of the cooling system.

2.2.2 Three-dimensional Computational Fluid Dynamics

The classical CFD has derived by solving the differential equations of Navier-Stokes for a fluid continuum with the method of finite volumes. The three basic equations that constitute the basis for today's CFD are the conservation of mass (2.2), momentum(2.3) and energy(2.4)-(2.5) integrated for a small control volume.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{v}) = 0 \tag{2.2}$$

$$\frac{\partial \rho \boldsymbol{v}}{\partial t} + \nabla (\rho \boldsymbol{v}\boldsymbol{v}) = -\nabla P + \nabla \cdot \boldsymbol{\tau} + \rho g \tag{2.3}$$

$$\frac{\partial \rho e_o}{\partial t} + \nabla (\rho \boldsymbol{v} e_o) = \rho \dot{q} - \nabla \cdot (P\boldsymbol{v}) + \nabla \cdot (k \nabla T) + \nabla \cdot (\boldsymbol{v} \cdot \boldsymbol{\tau}) + \rho g \boldsymbol{v} \tag{2.4}$$

$$\frac{\partial \rho h_o}{\partial t} + \nabla (\rho \boldsymbol{v} h_o) = \frac{\partial P}{\partial t} + \rho \dot{q} + \nabla \cdot (k \nabla T) + \nabla \cdot (\boldsymbol{v} \cdot \boldsymbol{\tau}) + \rho g \boldsymbol{v} \tag{2.5}$$

The energy equation has three different transport equations that could solve for, a transport equation for the total internal energy(2.4), the total enthalpy(2.5) or the temperature. The second is most often used by commercial CFD codes for segregated solvers. The biggest reason is the availability of data for the specific heat capacity at constant pressure c_P in most materials.

2.2.3 Turbulence theory and its effects in cooling system

Turbulence are instabilities in the flow that create scales which can interact with each other or with the main flow and eventually dissolve. The appearance of such phenomena has been broadly investigated as a result of the Reynolds number. Reynolds number is the ratio of the inertia to the viscous forces in a flow and is given for a certain diameter D as $Re = (\rho\, u\, D)/\mu$. According to White[12] in straight pipe flows the laminar region is observed at values of Reynolds number under 2300. In a cooling system there can be observed numbers as high as $Re = 20000$. Turbulence can be beneficial since it allows mixing along the cross-section, maintaining a more uniform temperature distribution and avoiding the creation of a warm fluid layer around the hot wall surface. This warm layer could act as an insulation layer that reduces heat transfer and possibly causes local boiling. The use of antifreeze promotes the return to laminar flow, which creates the need for higher mass flows. To accurately identify the turbulence effect the domain need to resolve scales of micro-meter (μm) size. A more broad approach to estimate turbulence is by defining equations that model the characteristics of turbulence.

RANS

The Reynolds Averaged Navier Stokes method derives from the split of the velocity variable to the time-averaged velocity and its transient, highly fluctuating, part. In incompressible flows the modeling focuses on the averaged correlation between fluctuating velocities $\overline{v_i' v_j'}$ named after their creator as Reynolds stresses. They can be substituted in the derived averaged equations (eq. (2.6)) from a stress tensor that is essentially the effect of turbulence in the mean flow.

$$\frac{\partial \rho \overline{\boldsymbol{v}}}{\partial t} + \nabla (\rho \overline{\boldsymbol{v}}\overline{\boldsymbol{v}}) = -\nabla \overline{P} + \nabla \cdot \overline{\boldsymbol{\tau}} - \nabla \cdot \left(\rho\, \overline{\boldsymbol{v'}\boldsymbol{v'}}\right) + \rho g \tag{2.6}$$

To define them two methods have been investigated. The first is to solve a transport equation for each of the 9 Reynolds stress terms. After obtaining equations for them, sevaral terms need closure and are modeled (Reynolds Stresses Model - RSM). This method increases the computational cost greatly. The second method is modeling the Reynolds stresses. The model that is most widely used is the Boussinesq approximation (eq. (2.7)), which tries to model the effect of turbulence according to the newtonian viscous stresses (eddy viscosity models). Continuity of the mean flow will cancel out the effect of the fluctuations in the normal direction, so another term that comes from the sum of the normal stresses is added. It is the turbulent kinetic energy k or the sum of flactuating velocities' energy $k = 0.5\sqrt{\overline{v_1^2} + \overline{v_2^2} + \overline{v_3^2}}$. In this approximation its effect is isotropically splitted to each direction. Lastly, accurate modeling is needed for the transport coefficient of turbulence, that is called turbulence viscosity ν_t corresponding to the kinematic viscosity in viscous stresses.

$$\overline{v_i' v_j'} = -\nu_t \left(\frac{\partial \overline{v_i}}{\partial x_j} + \frac{\partial \overline{v_j}}{\partial x_i} \right) + \frac{2}{3} \delta_{ij} k \tag{2.7}$$

Turbulence methods: k-epsilon vs k-omega SST

The two-equations methods that try to model the behaviour of the turbulent viscosity, focus on finding its correlation to three properties of turbulence that have been investigated: the turbulent kinetic energy k, the energy dissipation rate ϵ and the specific dissipation rate ω. The energy dissipation rate shows the rate that kinetic energy of the turbulent scales is transferred to smaller scales till eventually dissolving to heat. The specific dissipation rate ω is proportional to the ratio of k/ϵ. Transport equations have been analyzed for these three properties and can be solved together with the Navier-Stokes equations to determine the flow.[1]

The turbulent viscosity is then given by $\nu_t = C_\mu \frac{k^2}{\epsilon}$ or $\nu_t = \frac{k}{\omega}$, depending on which two of the three properties are solved for. More advanced models incorporate cases in the flow where the modeling of the eddy viscosity is done by different models per region or with the use of damping coefficients to reduce the effects of the modeled turbulence very close to the wall or in the free-stream regions. One example of the models that are region dependent is the SST model (Shear-Stress Transport model) that is changing from k-ω near the wall to k-ϵ far from it. As an example the modeled k-equation is presented in equation (2.8).[2]

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial\left(\rho k \overline{v_i}\right)}{\partial x_i} = -\rho\overline{v_i v_j}\frac{\partial v_j}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + \beta g_i \frac{\mu_t}{Pr_t}\frac{\partial T}{\partial x_i} - \rho\epsilon\left\{OR\left(\beta^* k\,\omega\right)\right\} \tag{2.8}$$

3 Methodology

3.1 General method for coupling 1D-3D

3.1.1 Geometry treatment

The complete cooling system is available by the department in 3D CAD files. These design files can represent the solid parts of the pipes, connections, valves and other components that consist the cooling system. After selecting the parts of the cooling system that will be investigated, the domain of the working fluid needs to be obtained in CAD file. In order to obtain it, a volume body is extracted that is surrounded from the inner walls of the pipes and the chosen inlet and outlet surfaces. This investigation is focusing on the flow behaviour of pipes, flow splits and bends and will test how they perform in 1D and 3D. The system of pipes can be split to different parts - for example regions that they are straight and others that have curves in the geometry - and all of these CAD geometries will be translated to suitable 1D blocks for the GT-SUITE model. Further manipulation on the CAD is needed for the connections of the parts, because a hose or a connector starts by covering the end of the previous subsystem in order to ensure the system is sealed. Therefore the exact point that the one part ends and the next one starts needs to be explicitly set.

A simplification to improve accuracy in 1D and simulation time in 3D is to change the real geometry of a hose that has a converging entry to a constant diameter as the rest of the system, thus reducing the error of the geometry translation to 1D and reducing the need of very fine mesh for 3D simulations. In GT-SUITE the blocks can represent a straight pipe with constant radius, a curved pipe with bend angles and varying radius, a flow splitter or a mitter bend. When a reduction or expansion of radius in the geometry occurs in a very small length compared to the discretization length its effect in the flow can be neglected. As a result, the radius is simplified to the same radius as the rest of the pipe, thus not allowing variations in the discretized length between nodes. In some of the cases in which such regions were not neglected, it was observed incorrect translation to an 1D block. In 3D, this part can be either neglected or included. The most often reason to want such a complex region neglected in a 3D simulation is the computational costs and the instabilities that can occur if the mesh lacks refinement in the region.

3.1.2 Configuring circuit of the 1D simulation

The tool that is offered by GT-SUITE to translate a fluid volume domain in 3D to a 1D block is called GEM3D. The CAD files that can be loaded are preferably loaded in .stl format. They can include all the models of hoses, flow splitters, mitter bends and quick-connectors. Each part can be translated to each equivalent block from the software, by reading the details of the geometry like bends, diameter variations and inlet-outlet port directions. The 1D block that is created is also visualized in 3D by the software to inform the user what is the geometry that represents. The user is offered the option to intervene to the various settings that are created from the tool, if the translation does not correspond to the actual 3D geometry.

There are further settings that are needed by the 1D software. At first, the initial state of the fluid inside the block needs to be set, to assign it properties like type of fluid, initial temperature and initial pressure. Moreover, the heat rejection through the wall can be set by the user. In cases that experimental data for pressure drop or heat transfer exist, the translated 1D block is set explicitly to provide the correct heat transfer or for the pressure drop is set frictionless and a different instantaneous block applies the pressure drop. The software allows for plotting in different locations inside a block or the extraction of signals to observe a variable's value in certain positions of the block. Subsystems with very complex geometry or flow physics such as pumps, heat exchangers, water jacket and others in 1D can be simplified to give the pressure difference or the heat transfer as they are expected to have it by experimental data. There are also more advanced models for modeling air cooling in air to water heat exchangers that are recommended by the software.

The boundary conditions for the cooling system will be set constant in time. A constant speed will be given to the shaft of the pump, simulating the case of running the engine at a constant engine speed. The solution will be time dependent and eventually converge to a solution for pressures and mass flows in the blocks. An implicit 1D solver for CFD is used to solve the Navier-Stokes equations. The time step can be limited by Courant number in time-dependent boundary conditions but for constant boundary conditions it just need to satisfy convergence. The value of 0.1 s is given as time step with inner iterations per time-step to reach up to maximum 200 or break if the desirable values for residuals have achieved. In the last case that was be investigated a transient boundary change in a valve was introduced.

For coupling a part of the 1D circuit with 3D, two different type of connections in the 1D model are needed. The first that is named as *CFDInterface* defines where a break in the system occurs and a connection with a 3D CFD simulation is placed. For the connection in an inlet or outlet port, the block connection that is offered to set the properties of it is called *CFDConn.* Available options are i) a mass flow , ii) a velocity and a iii) pressure interface. The mass flow and velocity interfaces are transferring the mass flow and velocity values to the 3D boundary and they expect the 3D to transfer the pressure value averaged in a volume region near the boundary. The pressure interface is used mostly as a pressure outlet in 3D with value given by 1D connection, while 1D reads the mass flow or the velocity at the 3D surface of the boundary (the user can select between the two). Two options are offered for the type of fluid that can be coupled between STAR-CCM+ and GT-SUITE, a multicomponent gas that is usually air and/or combustion products or a single-phase, constant composition liquid. The fluid properties translation from the 3D to the 1D is given by the blocks *CFDSpecies*,*CFDSpeciesMap* and *CFDDefinedSpeciesMap*, where different components or phases that may exist in 1D can be translated to a more simplified fluid in 3D. For example, existance of gas phase in 1D is taken into account and translated to liquid and the total mass flow of the two fluids will be given to the 3D simulation.

One important setting that needs to be set in *CFDConn* is the pressure relaxation time, which is a time span that when transferring of a pressure value between 1D and 3D system occurs, the new pressure is averaged with the previous pressure value so it will have a damping effect of any sudden pressure changes. This option is given to minimize continuity inconsistencies when a new pressure is given and to assure stability in the coupled system and convergence to the correct value in the 1D system.

The volume parts of the 1D system will be replaced by a single block that is called *CFDComponent.* The user needs to set when the coupling will occur after the 1D initialization and stabilization in a relatively constant result. This first period is named uncoupled period. Afterwards the coupling will start but for a chosen period the system will be one-way coupled, with only 1D sending boundary data to the 3D and ignoring the ones that 3D sends back. At a certain moment, two-way coupled simulation will start running and boundary values will be transferred reciprocally for the rest simulation time. The total simulation time should be enough to approach steady state behaviour. In this 1D block that will represent the 3D region, the user will also give a name of the various components in the 1D circuit that will be substituted by a 3D domain.[4][5][10]

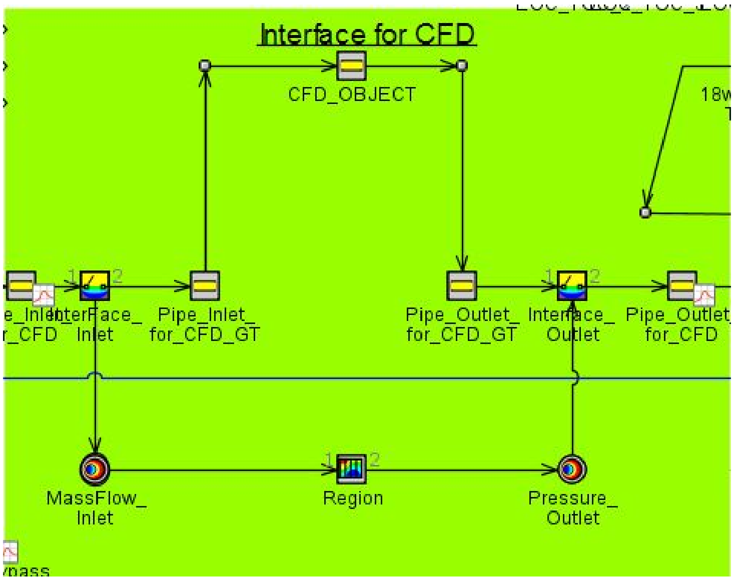


Figure 3.1: *The model to substitute three straight pipes in 1D with a 3D simulation.*

3.1.3 Preparing the 3D CFD for coupling

In the 3D software, the fluid domain is imported in .stp (STEP) format. The different geometry parts that represent a fluid volume are assigned to corresponding regions and the surfaces are assigned to the corresponding boundaries. Usually inlets are mass flow or velocity inlets and outlets are selected as pressure outlets. Afterwards, the meshing of the region is required. The mesh is created automatically with an automated mesh feature by STAR-CCM+. The recommendation from theory to create the volume mesh is to keep y+ values at wall boundaries around the corresponding values by the different turbulence models. The wall y+ values in the recommended models for RANS method should be between the values of 1 and 30. In the problems that are dealing with flows within cooling system components the average size of a cell is suggested to be between the values 1 and 5 mm. The prism layer feature is setting a hexagonal mesh near a wall boundary and chooses the number of layers of mesh, the thickness near the wall of the total layers as well the stretching factor and the model for growth.

In the rest of the domain the user has 4 available types of meshes, the tetrahedral, hexahedral, polyhedral or a type of a cylinder mesh specialized for domains with circular cross-sections. The nature of the domain can be a factor in choosing the most suitable mesh. In pipes the domain has a circular cross-section. Therefore the cylinder mesh is the most relevant to the cases with cooling system components. It creates a polygon mesh in the inlet and outlet of the part and then it superimposes it along the domain, creating a combination of a polyhedral and hexahedral mesh. This mesh allows the biggest dimension of a cell to follow the estimated flow direction, thus increasing also the maximum time-step limit. In parts that have more complexed domain the polyhedral mesh will be used. It is composed by tetrahedral cells with the neighbouring cells merging to a bigger cell, thus helping to decrease the memory and calculations.

The coolant in STAR-CCM+ is represented by a single-phase liquid with user determined density, specific heat and thermal conductivity. They can be either given as constants or as functions of other thermodynamic properties(Temperature, Pressure). The solver that is used in incompressible flows is a segregated flow solver for velocity (momentum equations) and pressure (continuity equation). If there is energy transport or exchange in the system, an additional energy equation will be solved in a segregated form, with the option of either solving for the temperature or the enthalpy transport equation.
The domain is initialized by running a steady state simulation with boundary conditions that are obtained manually from a simulation for the single 1D system. As a result instabilities and incorrect solution that are caused by incorrectly initialized domains are not propagating in the unsteady simulation where the two solvers are coupled. The coupling is offered only with an implicit unsteady time discretization of a liquid with constant composition. To connect the GT-SUITE simulation to the STAR-CCM+ run, a directory of a specialized

library for coupling that is available in the GT-SUITE installation is given to the settings for co-simulation in STAR-CCM+. This permits the coupling to be controlled by STAR-CCM+ which eventually becomes the master host that launches the other software and runs it. For the coupling the two software were available for Linux operating system. Moreover the model of GT-SUITE that is used is pre-processed for simulation, creating a read file in .dat format that will be used directly by STAR-CCM+. The same pre-processing could be also achieved directly from the master software, but would take longer to complete it. By performing the pre-process of the 1D simulation in its software environment, the user has the advantage that the complete 1D system and its sub-components can be created in a different server/computer and then only show the directory of these files to the server of the master software. Three files are required in the same directory, the read file created in .dat format, the model file .gtm and a file to setup post-processing of the results in .sim format.

To extract the results of the 3D simulation a report of a surface average at the interface of the boundary is computed to obtain the properties of the fluid, such as pressure, temperature, etc. Furthermore, the mass flow through the interface is another report option that can be extracted. A file is created that contains all the variables for the different interfaces and the time they were recorded. For visualization, planes that are parallel to the flow can be plotted to show the variation of pressure or velocity magnitude, while streamlines that are originating from the inlet can show the direction of the flow and re-circulation regions.
[8]

3.1.4 Choosing turbulence models and boundary conditions for 3D

To perform a simulation with 3D CFD for cooling systems, the turbulence is approached with the RANS method. The advantage of this method is on the less strict limitations for time-step and cells size. More specifically, in the methods that want to resolve a part of the turbulence from the Navier-Stokes(LES, DES, etc) the cell size needs to be lower than a maximum value near the walls. The limitaiton is valid for all streamwise, spanwise and normal to the wall direction. Moreover, after creating the mesh the time-step has to be adjusted accordingly to satisfy the local Courant–Friedrichs–Lewy number of the mesh. It accounts for the maximum cell jumps the flow is allowed to perform within a time-step. The Courant value limit is 1 for explicit time-discretization and 5 for implicit.) The time-averaging that takes place in RANS permits to increase the time-step in unsteady RANS for the quasi-state cases that were investigated in this project. For cases that exist highly unsteady boundary conditions URANS needs also to adjust the time-step and/or the cells size to the Courant limitations.

In STAR-CCM+ two different turbulence modeling methods are recommended: the realizable k-epsilon method with two-layer all y^+ wall treatment and the k-omega SST with all y^+ wall treatment. There will be performed a comparison of the two recommended models in different systems and for different operating points, but the unavailability of experimental data will limit the conclusions of this comparison.

The boundary conditions that are used in cooling systems are given mass flows in the inlets and given pressure in the outlets. For an inlet, coupling permits only a uniform profile with direction normal to the inlet surface. This neglects the turbulence in the flow and the boundary layer on the walls. To counteract this the user can give a prescribed constant value of turbulence at the inlet, but that creates the problem that turbulence does not have a uniform profile. Nonetheless, a better initialization of turbulence can help accomplish the formation of the boundary layer a bit earlier than for a flow with zero turbulence. When the boundary layer is formed further away from the inlet, the excess pressure drop, that is the result of irrotational flow in the entrance region, increases. Lastly, there is small knowledge of the average kinetic energy or turbulence length in the inlet, while no experimental data were available.

When mass flow boundary condition is set in an inlet and pressure in an outlet for the 3D simulation, that correspondingly creates an opening in the 1D closed circuit. The interface where the pressure is given to 3D, the mass flow value of the coolant in 3D is transferred to the 1D simulation creating effectively a mass flow inlet for that system. That system would close with a pressure outlet boundary, which is given by the pressure in the inlet of the 3D that is transferred to the 1D. As a result from a closed 1D system, one open 1D and one open 3D are created with same concept in their boundary conditions.

3.1.5 Post-processing of the results

In this investigation it is really important to create plots that show the flow properties as they were calculated by the two different software. A post-processing tool in python code was created with the scope to automate the process of extracting the data of the different software, load them correctly and plot them superimposed. Initially, the data are extracted in a file by the STAR-CCM+ software at the end of the simulation. It contains the flow properties such as velocities, temperatures, etc that are saved per time-step in the different inlet/outlet ports or other possible locations that can be set by the user. For GT-SUITE, an object file is created, that is opened by a separate post-process application offered by the software and called GT-POST. This presents graphically the flow properties in the circuit and the different components of the cooling system, while also contains all the plots and convergence data that the user set to be recorded. These data can be either exported manually for each block to a corresponding file or group them together with a certain configuration tool and export the data of several locations to a single file.

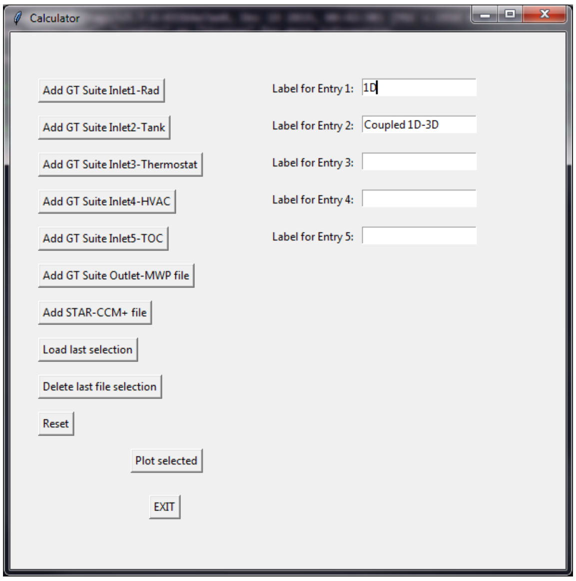


Figure 3.2: *Interactive GUI for automated post-processing.*

The post-process tool loads the data files by selecting the appropriate files through an interactive Graphical User Interface GUI. It has been created with the use of the python package tkinter. After choosing all the desired files with the data that will be plotted and write the appropriate plotting labels in the corresponding cells, a click button on the GUI reads the data and their variable names and activates a plotting function, that creates plots superimposed for the different GT-SUITE and/or STAR-CCM+ results for variables and locations interesting to the project.

3.2 Study cases description

Both software have a detailed tutorial to establish a co-simulation run. After completing all the steps, the project had the opportunity to perform a series of simulations. Initially, it was desired to define an optimised configuration for the coupling. With the investigation of each new case study, the complexity of the system to be modelled in 3D would increase. The cases are concluded with a modeling of a big part of the cooling system with 3D CFD and its behaviour would be compared to the results of the other study cases to identify trend patterns for the coupled simulations.

For this master thesis, a simplified version of the cooling system in a petrol engine of a Volvo Car model was developed, as presented in Figure 3.3. With blue is the flow in the cold parts of the system and red the hot. The flow from the pump is directed to the engine block, the oil cooling systems (EOC/TOC) and the EGR. At the engine block the flow splits to three different routes. The majority of the flow is regulated by

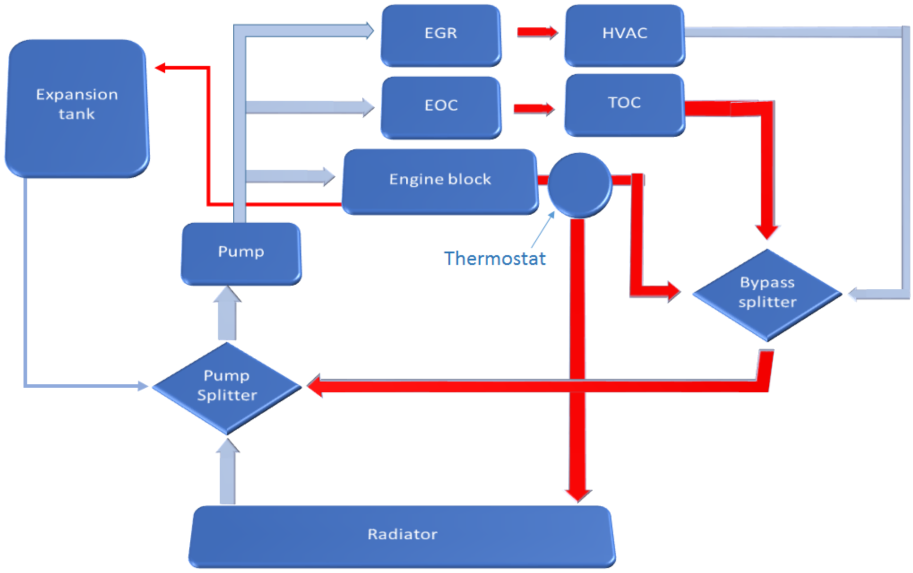


Figure 3.3: *The layout of cooling system used for this project.*

the thermostat, while a smaller part of it being directed to the expansion tank through a degas pipe. The latter will permit the separation of vapor or air in the system from the liquid coolant. The thermostat will decide if flow will go through the radiator. When the thermostat is open the majority of the hot water coming from the waterjacket will be cooled in the radiator, while a part of the flow will be directed back to the pump through a bypass. When the thermostat is closed all the hot water will be directed to the bypass, thus helping to warm-up the engine and regulating the coolant temperature in it. The coolant from the EGR goes to the climate system to provide heating (HVAC). After HVAC the flow will be re-connected in a 4way splitter with the flows from the bypass of the thermostat and EOC/TOC and it will be directed towards the pump. The exits of the flow from the expansion tank and the radiator are also directed towards the pump, where they merge with the bypass flow in another 4-way splitter, directly before the inlet of the pump.

3.2.1 Case 1 - Straight pipe

Figure 3.4: *Meshing of the straight pipe.*

The most simplified flow of a cooling system is through a straight pipe. It was desired to investigate a long pipe in 3D, to establish a working co-simulation with the rest system in 1D and to perform a series of simulations with different settings. The simplified geometry permits for a smaller refinement in the mesh, which will allow faster simulations. Therefor a big number of different parameter settings can be tested. The pipe was needed to be added externally to the system as a cooling system doesn't have pipes that have straight geometry in sufficient length. The extra part was added before the inlet of the engine block and it has length

0.28 m and diameter 2.8 cm.

	Coupling settings			3D convergence setting		
	Uncoupled period	One-way coupling period	Pressure relaxation	Time-step in 3D	Inner iterations	Continuity convergence criteria
Value #1	60 sec	1.5 sec	0.5 sec	0.05 sec	3	No criteria
Value #2	**80 sec**	**2.5 sec**	**1.0 sec**	0.1 sec	12	1.0 E-10
Value #3	100 sec	5 sec	4.0 sec	**0.2 sec**	20	**1.0 E-8**
Value #4	-	-	-	0.3 sec	-	5.0 E-8
Value #5	-	-	-	0.5 sec	-	-

Table 3.1: List of different settings that were tested for co-simulation at operating point of 2000 rpm. The values that were considered the baseline for the rest simulation are highlighted.

The parameters of coupling that were investigated are depicted in Table 3.1. Uncoupled period in seconds is the initial period of the co-simulation that 1D runs till it will reach a constant state. One-way coupling is the period in seconds that both simulations have been initiated, but the transfer of the boundary values is happening only from 1D towards 3D. This has the effect of a second initializing of the 3D domain with an implicit unsteady solver. The first initialization took place with a steady state solver. After these two periods the full coupling starts. The pressure relaxation setting that is offered in GT-SUITE was also investigated. Moreover, an investigation on the effect of the time-step in the 3D simulation of the straight pipe was performed. Finally, convergence settings for the 3D simulation were tested and more specifically the inner iterations between time-steps. The options were to give a different constant value of inner iterations, or a continuity criteria that would finish the iterations for that step if the continuity residual would be lower than the criteria.

3.2.2 Case 2 - Curved hose

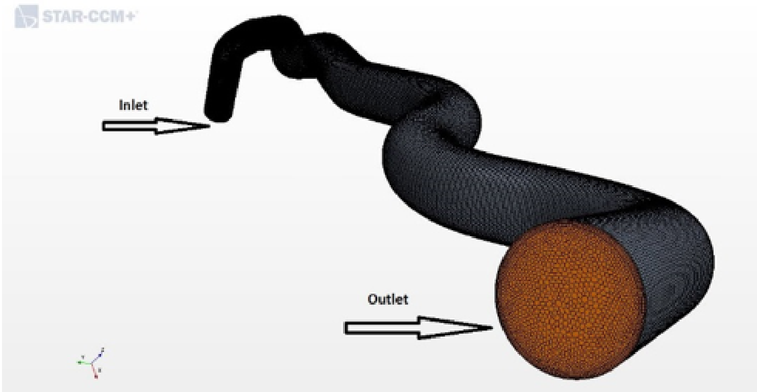


Figure 3.5: *Meshing of the curved hose.*

The second case was to investigate a curved hose with bends. The hose was selected to be the hose located between the thermostat at the engine block and the radiator. It contains a series of bends, while it has a total length of about 0.8 m and constant diameter of about 2.8 cm.

The coupling and simulation settings would follow the optimal result of case 1. Interesting topic of research for this case was the behaviour of different turbulent models. As mentioned before it was used the RANS k-epsilon and k-omega models, at their default settings by the software. The operating point of the cooling system was varied for three increasing engine speeds , 2000 - 4000 - 6000, increasing subsequently the mass flow provided by the pump. Meshing was adopted for each model differently in order to achieve proper convergence and be within the acceptable limits for cells adjacent to the wall.

3.2.3 Case 3 - Bypass flow split

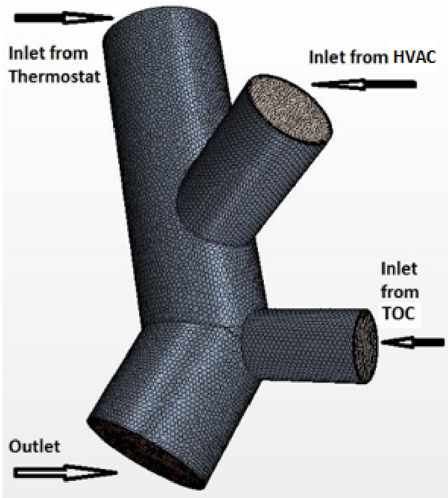


Figure 3.6: *Meshing of the bypass flow split.*

To increase the complexity of the problem and to identify the behaviour of the system when it is introduced additional interface ports, a flow split was modeled in 3D. It was selected to be the bypass split that has three inlets coming from the thermostat, the TOC and the HVAC and one outlet towards the pump splitter. The 1D is modeled with a block that decides on the expansion length and diameter for the different ports of the flow split. It also reads the total wall surface and volume of the CAD, in order to impose the 1D equations. For the meshing and turbulence model that would be used in 3D, it followed the suggestions deriving from case 2.

The additional inlets to the system would be given as mass flow inlets to the 3D configuration returning the pressure value to the 1D software. The outlet is still a pressure outlet. It is of interest to investigate if the behaviour of the flow in the ports would follow the same trend for all the ports.

3.2.4 Case 4 - Flow split before pump

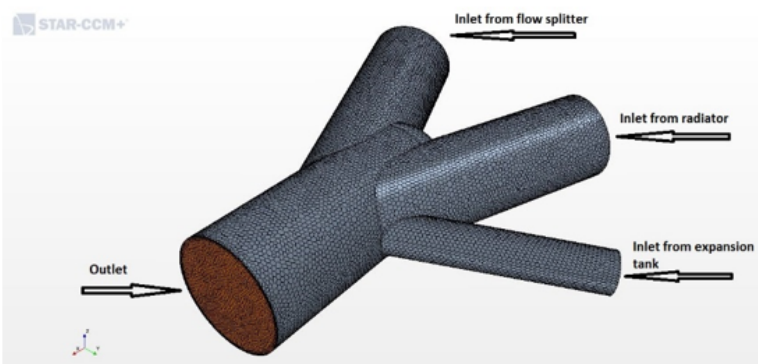


Figure 3.7: *Meshing of the splitter before the pump.*

The flow split before the pump doesn't introduce higher complexity to the system, but it belongs to a special case since it is located before the pump. When the cooling system is modeled in 1D, the block that provides the flow re-circulation in the closed system is the pump model. It reads the instantaneous pressure and provides the mass flow. For the models that are developed with GT-SUITE, it has been identified that the pump model introduces instabilities. These instabilities appear in transient simulations, for which the pump follows a pump curve from experimental data. The solver has dampening factors that would force the system to approach conservatively the solution. Otherwise at a certain pump speed, slight differences in mass flows can introduce non-physical pressure jumps in the system.

This case was investigated with two scenarios. For the first scenario, it was simulated a coupled case with the pump modeled in its standard form. The second scenario would substitute the pump model with a closed circuit boundary condition that imposes a pressure outlet before the pump, a mass flow inlet and periodic temperature. The values would be obtained by running the system in 1D with the standard pump model.

3.2.5 Case 5 - Part of the cooling system before the pump

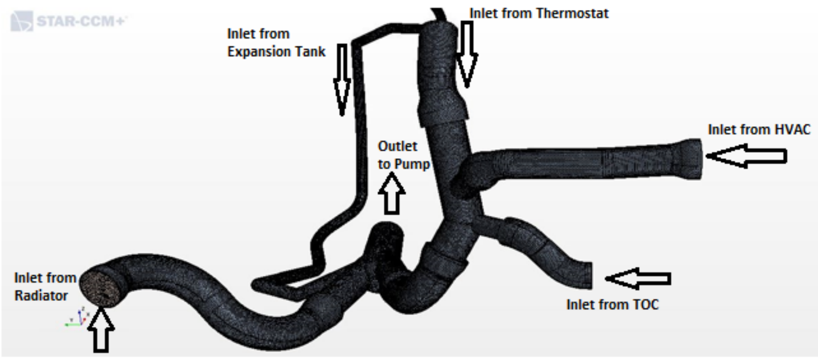


Figure 3.8: *Meshing of the part of the cooling system before the pump.*

The last case would be to model a big part of the system that would include only hoses and 4-way splitters. It would end in the flow split before the pump that is already modeled for case 4. The 4-way splitter of case 3 after the thermostat and the hoses connecting the splitters with their neighbouring components were included in the 3D model.

The method that will be used for the modeling of the pump in the 1D system would be according the conclusions of the case study on the case 4, flow split before pump. It will be shown that the modified boundary of the pump will be used. The results of the co-simulation will be compared with results from single 1D simulation. The inlet ports would be located in the outlets of the i) radiator, ii) expansion tank, iii) thermostat to bypass, iv) HVAC and v) TOC, while a single outlet port would be used in the inlet of the pump.

After establishing the simulation functionality, a more advanced case would be introduced with an instantaneous change on the position of the thermostat. It would be modeled both a case that the thermostat's position would be move from open flow towards the radiator to close, and the opposite. The instantaneous change of a valve's position requires instant adjustments to the flow as it is provided by the pump. A 1D simulation needs to run for the system with the standard pump model to obtain the mass flow and pressure before the pump that the system settles for each thermostat's position. Afterwards, the coupling of the system with the simplified boundary model for the pump and the change of thermostat position in the middle of the simulation would be simulated.

Part II
Results, Discussion and Conclusion

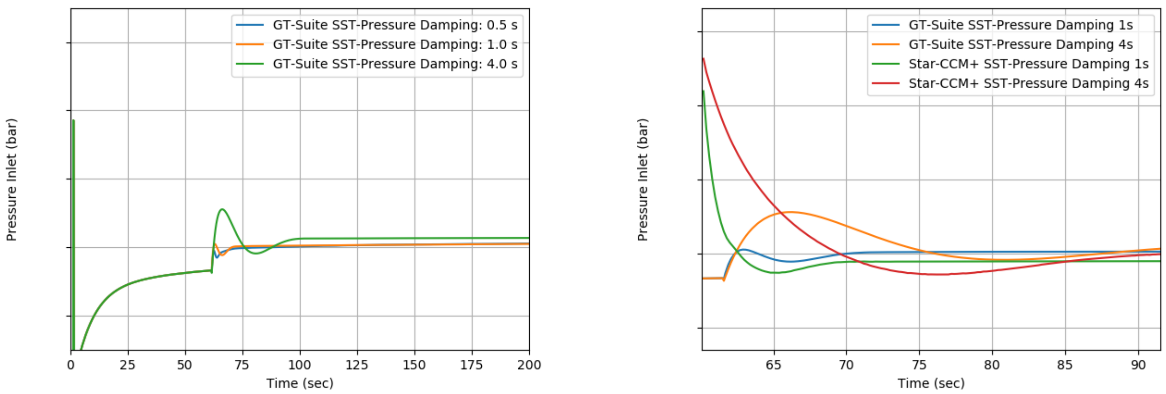
4 Results

4.1 Case 1 - Straight pipe

4.1.1 Coupling settings investigation

After configuring the 1D system to accommodate the coupling of the software, the 1D simulation was run uncoupled till convergence with engine speed set at 2000 rpm. The post-processing tool of GT-SUITE offers the mass flow inlet and pressure outlet values that were required to initialize the 3D domain. The co-simulation was tested with proper initialization, it finished without problems and the results were within the expected range.

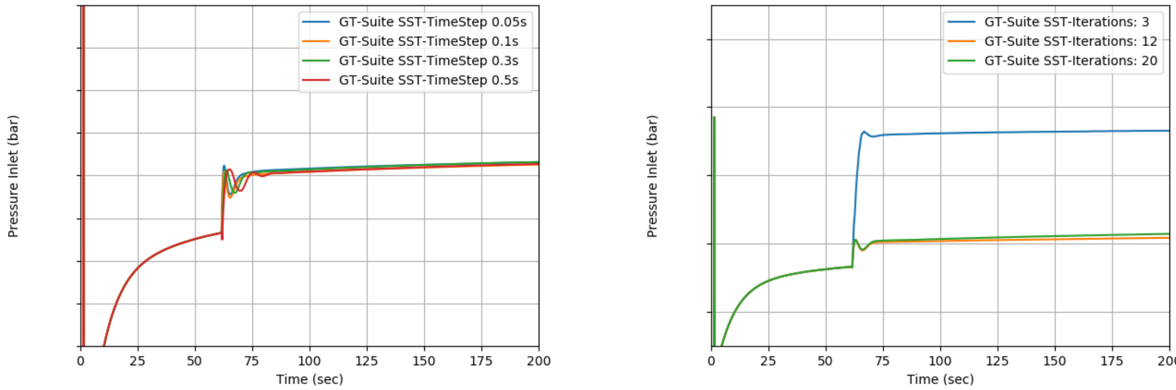
At next, a list of different settings were simulated with the engine speed at 2000 rpm as mentioned in Table 3.1. The results were plotted against the time, to compare the behaviour of the flow properties in the interfaces at the point of coupling till the system would converge. They were available both from the 1D and the 3D software. Mass flows, pressure and pressure drop between inlet and outlet were the focus of the investigation. The plots at Figures 4.1 and 4.2 present an example of the reaction of the system at the point of coupling when the co-simulation run with the different settings. From the rest simulations it is seen that uncoupled period didn't change the behaviour of the system. The one-way coupling period showed a correct initialization of the 3D domain at the point of two-way coupling. Higher values of one-way coupling did not account for improvement in initializing the 3D but increases the computational time since 3D simulation runs for longer simulation time. After a certain value of the one-way coupling period the pressures and mass flows are converging to the same coupling behaviour and end result. For smaller coupling times it is noticed that the reaction of the system at the point of two-way coupling is causing a slightly higher jump . Mass flows and pressures are the same in the end results.



(a) (b)

Figure 4.1: *a) Pressure at the inlet for different damping pressure values, b) Zoom-in at the coupling period and plot for results obtained by 1D and 3D.*

The pressure relaxation setting gives a weighted value for the pressure when it is imposed by the 1D or the 3D software. The weight happens with parameter with time units that act as a dampening factor. The results (Figure 4.1) showed that till moderate values the coupling point showed no instabilities, while large value showed a high jump in mass flows and pressures. The smaller the pressure damping the faster the system



(a) (b)

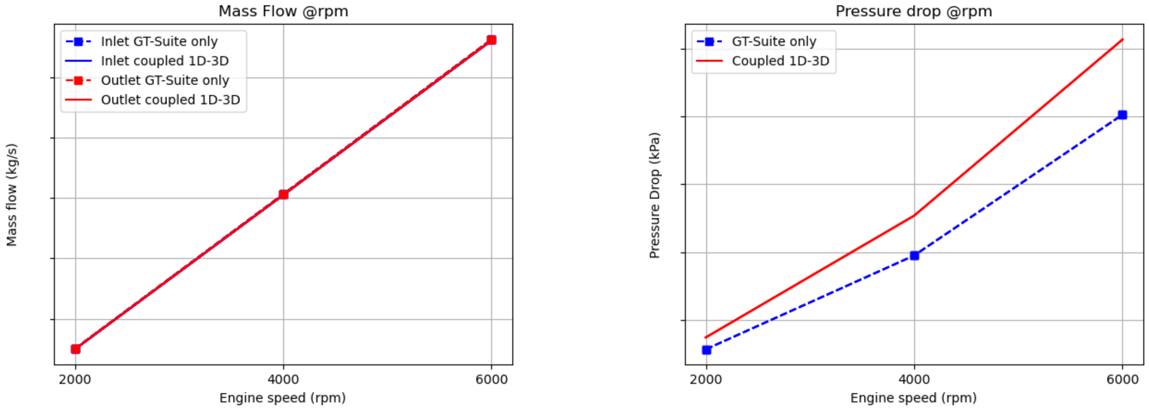
Figure 4.2: *Pressure at the inlet for different a) time-step in 3D and b) constant inner iteratins.*

reacted to the coupling point.

The rest of co-simulation settings that were tested, wanted to investigate convergence between time-steps in 3D CFD and how this affect the point of coupling and the result of the co-simulation. With time-step in 1D constant at 0.1 secs, 3D was given values from 0.05 to 0.5 secs. At this simulation it was used constant inner iterations. The results in Figure 4.2a showed that different time-steps gave the same result but the jump at the coupling point behaved differently. More specifically, it is shown that lower time-steps have bigger spike but also converge faster to the solution. The highest time-step converges the slowest with the oscillation vanishing later than the rest time-steps. Moreover, a constant number of inner iterations was tested and the results for pressure and mass flow are shown in Figure 4.2b. Too few inner iterations cause a strong inaccuracy of the result. Above a value of 10 inner iterations the system was behaving similarly, but for further simulations a minimum acceptable number of iterations could be affected by under-relaxation factors in the 3D solvers or the number of cells. A better convergence method was then used with a continuity criteria and a minimum number of inner iterations. The results, that were also compared with the constant inner iterations, showed similar behaviour with a moderate value of inner iterations. The simulation time was highly decreased though, since for running much after the coupling point where the co-simulation was stable, it was using much less inner iterations. The criteria value that was used was for the continuity residual was between $5 \cdot 10^{-10}$ and $5 \cdot 10^{-8}$. For smaller continuity criteria, the simulation time was bigger.

4.1.2 Co-simulation of straight pipe at different engine speeds

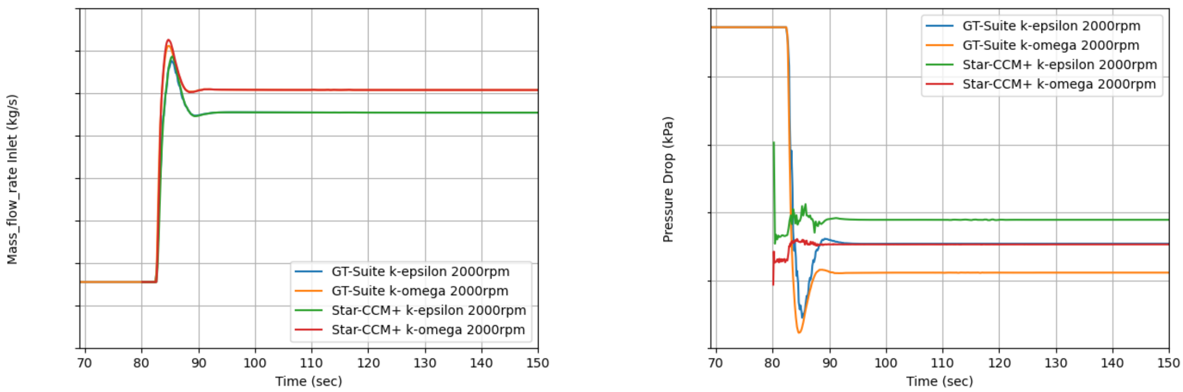
After establishing a validated and relatively fast method for the co-simulation of the straight pipe, a variation in the operating point of the cooling system was performed. By changing the engine speed from 2000 to 4000 and 6000 rpm, the mechanical pump would be driven with doubled and tripled speed. The mass flow and pressure jump would adjust to accommodate the new operating point. Eventually, the straight pipe of the coupled section of the cooling system would see higher mass flow and pressure drop. The results are depicted in Figure 4.3 for the end of the coupled simulation, in comparison with the converged 1D results. As in the previous plots, the values are adjusted for confidentiality. The mass flow and pressure of the system seem to be the same, but the slight variation in pressures between 1D and co-simulation causes a variation in pressure drops of the straight pipe. The pressure drop when the straight pipe is simulated in 3D is higher than when it is in 1D. The difference between the pressure drops increases gradually for the higher engine speed. The maximum pressure drop difference between the two methods is around 1 kPa at 6000 rpm. At cooling system evaluations, pressure drops variations of this magnitude are considered of low significance.



(a) (b)

Figure 4.3: *a) Mass flows at inlet and outlet of the system for different engine speeds of Case 1, b) Pressure drop of co-simulation compared to 1D for different engine speeds of Case 1.*

4.2 Case 2 - Curved hose



(a) (b)

Figure 4.4: *a) Mass flow at inlet of Case 2 for different turbulence models at 2000 rpm, b) Pressure drop for this configuration. Results obtained by 1D and 3D.*

As mentioned in the methodology, this case models a hose before the radiator in 3D with the rest cooling system in 1D. The several bends along the hose cause several direction changes of the flow. Even though the diameter is constant, the bends can cause instability in the flow and creation of high turbulent regions. An investigation on the turbulence models was performed. The realizable k-epsilon and the k-omega SST models have been tested for the three different operating points. The table 4.1 presents the different configurations of mesh that were used for each model and operating point, as well the running time for running in 6 cores. The k-omega model required high refinement in order to converge, while furthermore a time-step decrease was required. The k-epsilon needed a slightly more refined mesh for the case with the higher mass flow at engine speed 6000rpm. The results are depicted both against time (Figures 4.4 and 4.5) and at the different operating points. At first the behaviour of the co-simulation at the point of two-way coupling was not abrupt and both models showed the same trend in increasing the mass flow and decreasing the pressure drop compared to the 1D values. The results of the co-simulations with the two different models were close, whereas the 1D model results showed higher variation in the results and increased pressure drop.

	2000 rpm		4000 rpm		6000 rpm	
Turbulence model	$k-\epsilon$	$k-\omega$	$k-\epsilon$	$k-\omega$	$k-\epsilon$	$k-\omega$
Number of cells	380 k	1 m	380 k	1 m	450 k	1 m
Timestep (s)	0.2	0.1	0.2	0.1	0.2	0.05
Number of cores	8 cores		8 cores		8 cores	
Runtime	17 mins	12 hrs	20 mins	21 hrs	35 mins	27 hrs

Table 4.1: Simulation settings and compuational time for case 2.

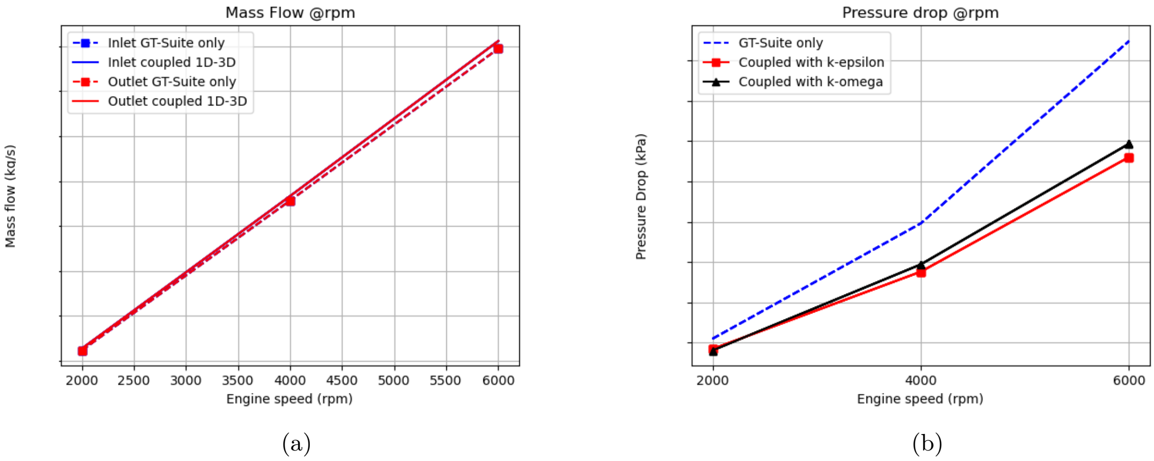


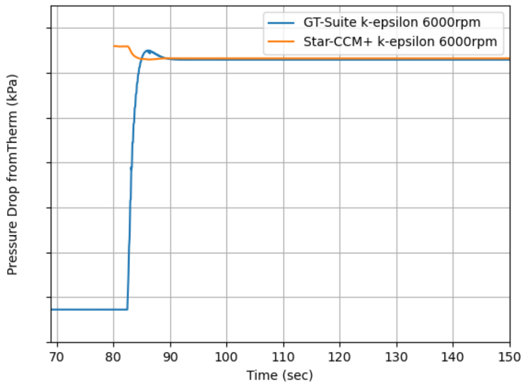
Figure 4.5: *a) Mass flows at inlet and outlet of the system for different engine speeds of Case 2, b) Pressure drop of co-simulations with different turbulence models compared to 1D for different engine speeds of Case 1.*

4.3 Case 3 - Bypass Flow split

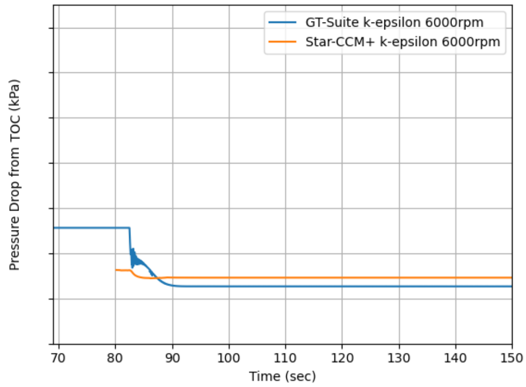
A different type of complexity was introduced to the co-simulation approach in the case 3. It was modeled in 3D a system with more than two interfaces. In the CAD it is interesting that the different inlets and outlets have different diameters. Only 3D polymesh type of mesh could be created by the automated mesh. The simulation was performed at the three different operating points, same as the previous cases. After deciding on the turbulence model and its advantages, the rest co-simulations were performed with k-epsilon model. At Table 4.2 the number of cells and the computational time are presented at each different engine speed. It was a fast simulation since it is a small component in the cooling system. The results followed the clear trends of the previous co-simulations with a small jump at the point of two-way coupling. In Figure 4.6 the results are plotted over time, for the data that were received on the interfaces by the 1D and the 3D software. The coupling point showed same behaviour as the previous simulations. Moreover the end results are compared to the converged values of the single 1D simulation at the different engine speed in Figure 4.7. While the mass flows and the pressure in the outlet remained at the same values, the pressures in the inlets showed two behaviours. It was increased for the flow coming from the Thermostat and HVAC, while it was decreased for the flow coming from TOC. The pressure drops of each port, that is the pressure difference of the port to the outlet, by definition, reacted same as the pressures.

	2000 rpm	4000 rpm	6000 rpm
Turbulence model	$k-\epsilon$	$k-\epsilon$	$k-\epsilon$
Number of cells	110 k	110 k	150 k
Timestep (s)	0.2	0.2	0.2
Number of cores	24	24	24
Runtime	12 hrs	10 mins	1.5 hrs

Table 4.2: Simulation settings and compuational time for case 3.

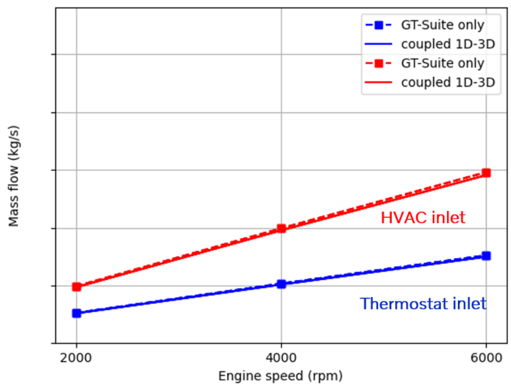


(a)

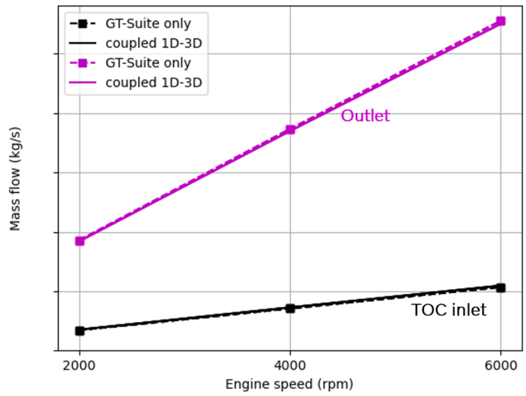


(b)

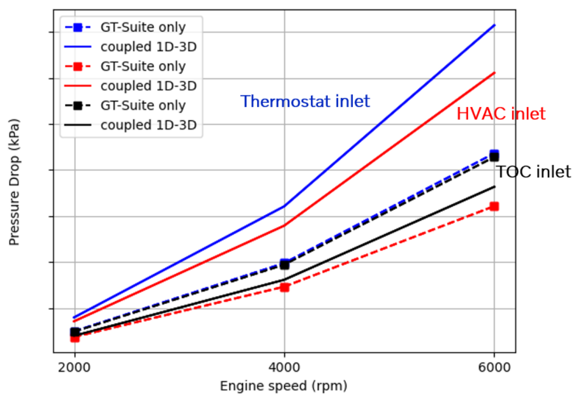
Figure 4.6: *Pressure drop for inlet from a)thermostat and b) HVAC of Case 3 at 6000 rpm. Results obtained by 1D and 3D.*



(a)



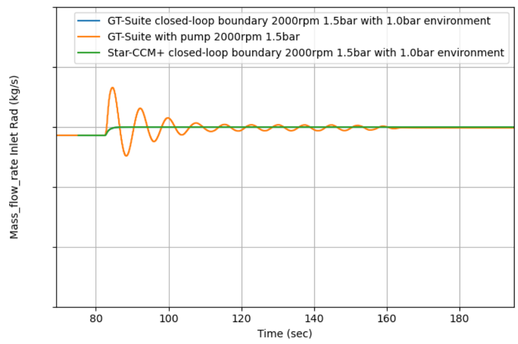
(b)



(c)

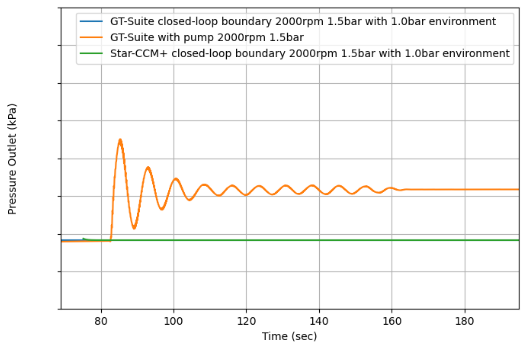
Figure 4.7: *a) Mass flows at inlet from thermostat and HVAC ,b) at inlet from TOC and outlet. c) Pressure drop for each inlet of the system for different engine speeds of Case 3.*

4.4 Case 4 - Flow split before pump



(a)

(b)



(c)

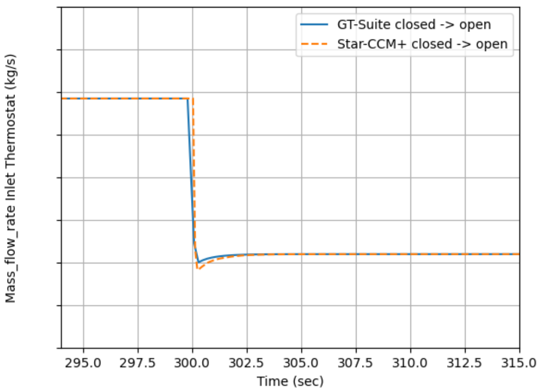
(d)

Figure 4.8: *Co-simulations' results for Case 4 with standard or simplified (closed-loop boundary) pump model. Mass flow at a) inlet from radiator and b) outlet. c) Pressure at the outlet and d) pressure drop from radiator inlet. The results of the simplified model are obtained both from 1D and 3D.*

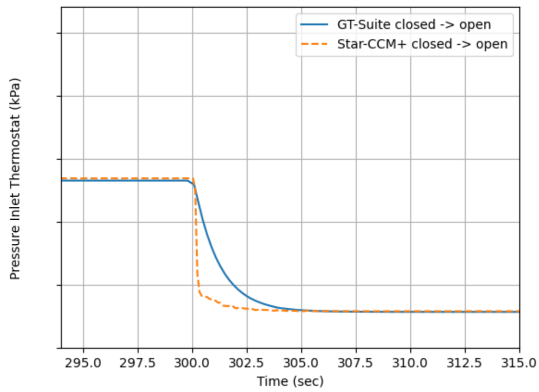
While this problem didn't introduce any flow complexity in the 3D, it was interesting to perform the co-simulation from the perspective of the cooling system as a closed system. Parameters that characterize the 1D system are the initial pressure and the engine speed. Two different initial pressures were used for simulations. The co-simulation was performed as previously at three operating points. The 1D model was the same as for the previous cases. Moreover, the flow split was modeled in 1D with the same method as in case 3, while in 3D it was meshed with the same settings as for that case. The results showed inability to stabilize the solution after the coupling point. For lower speed at 2000 rpm the oscillations that caused from two-way coupling were dampened. Higher oscillations occured for the 4000 rpm, that never dampened completely. By increasing the initial pressure - at calm state - of the closed circuit by 50%, with engine speed at 2000 rpm, these oscillations were eventually dampened after time (Figure 4.8. The case of 6000 rpm failed to converge after the initial oscillations.

The different approach mentioned in the methodology chapter for the pump boundary conditions was introduced. The higher initial pressure was retained for these simulations. At 2000 rpm the solution converged to the same result but with zero oscillation(Figure 4.8). The simulations converged for co-simulations at 4000 rpm and 6000 rpm. The coupling point behaviour that emerged from the new model for the pump was quite smooth and similar for all the operating points. The results were sufficient to proceed with the pump model as a boundary condition in case 5.

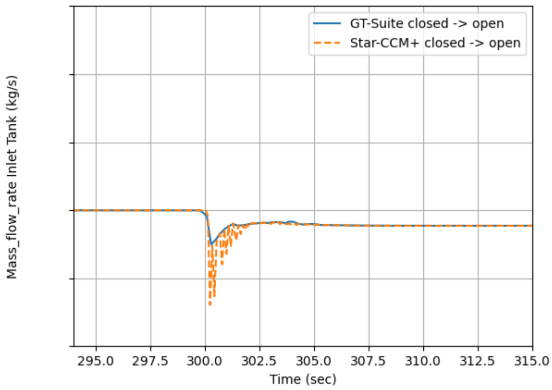
4.5 Case 5 - Part of the cooling system before the pump



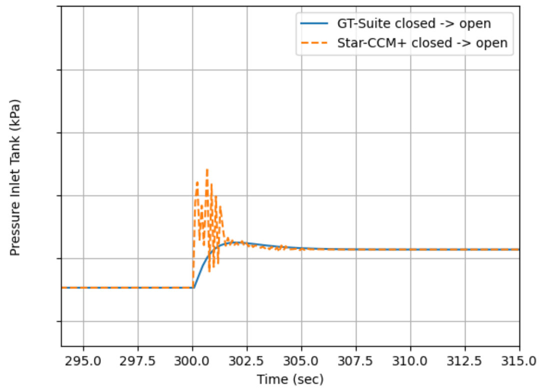
(a)



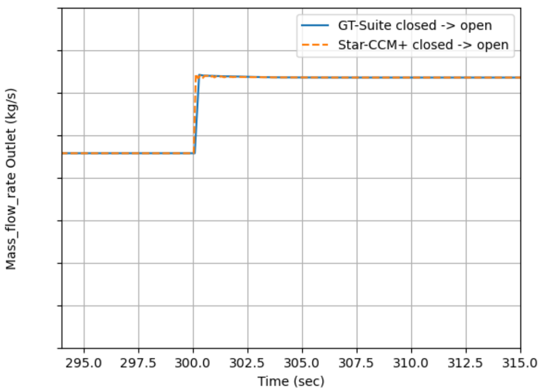
(b)



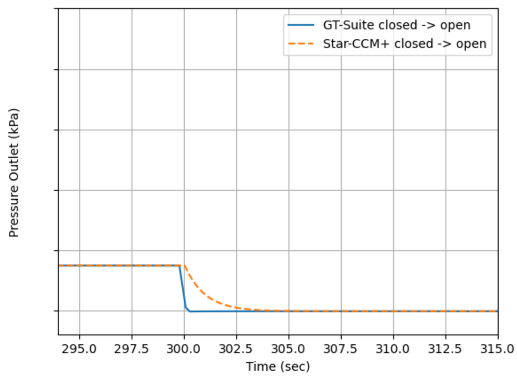
(c)



(d)



(e)



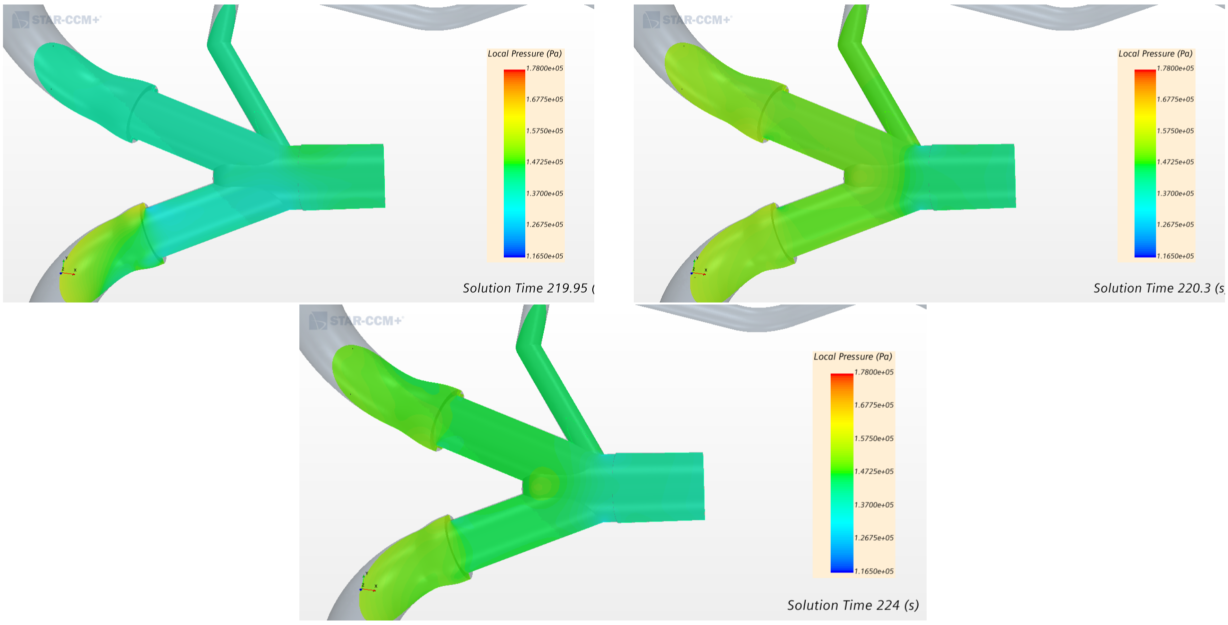
(f)

Figure 4.9: *Behaviour of the co-simulation at the point (t=300s) of changing the position of thermostat from closed to open. Mass flows and pressures for a) and b) inlet from thermostat, c) and d) inlet from expansion tank, e) and f) outlet to pump .*

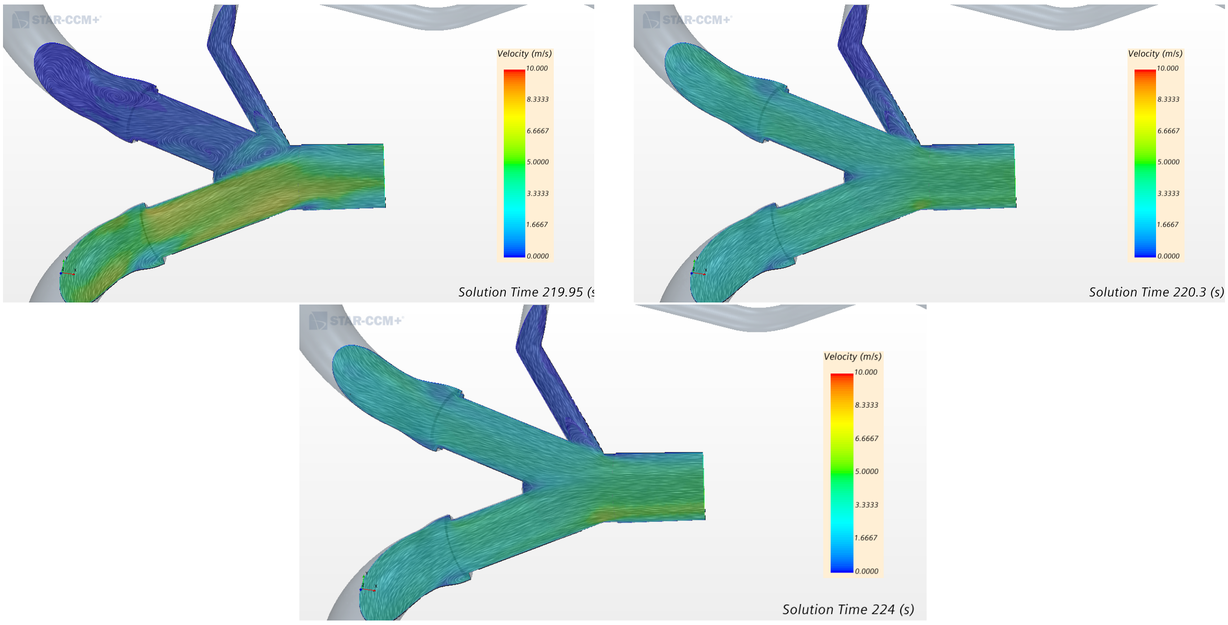
The single 3D simulation with the bigger part behaved according to the previous simulations. Same mesh settings and turbulence model were sufficient to converge the steady-state simulation for initialiazing the domain. The simulation was performed though with lower time-step than the previous cases at 0.05 s for the 3D simulation, while 1D implicit solver kept the same as before at 0.1 s. Furthermore, it was performed for both cases of open and closed thermostat, with the engine speed constant at 4000 rpm. The values were taken from the 1D simulation. The increased initial pressure was used for this simulation also, as in case 4. At the point of two-way coupling the system behaved as for case 4. At the new point of interest, the moment of changing thermostat position at $t = 300$ s, the system showed smooth adjustment to the sudden change of values in the boundaries of the pump, as per Figure 4.9. The plots show an example of mass flow and pressure behaviour over time, as they emerge from the 1D (GT) and 3D (STAR-CCM+) data for the inlets to the 3D coming from the thermostat and the expansion tank, as well the outlet interface of the 3D that is the inlet to the pump. While the mass flows had quite similar data, for the pressure it can be understood that in the inlet the 3D adjusts faster to the new operating and the 1D follows it. The opposite is presented for the pressure in the outlet, where the 1D system changes momentarily to the new pressure introduced by the boundary and the 3D slower adjusts to it.

Pictures of the flow in the 3D were also obtained at the moment of changing the thermostat's position. For the presented case the thermostat is closed and then it opens at $t = 220$ s. Figure 4.11 present the evolution of the inner flow in the 3D part of the co-simulation. The streamlines show the high turbulent region at the bend between the two flow splits. Moreover, transient changes in the flow appear in the hose coming from the tank. A big recirculation region near the point of connection of the flow coming from the tank hose with the rest flow in the last flow splitter appears when the thermostat is closed and there is no flow coming from the radiator.

Lastly, in Figure 4.12 the result of the co-simulation is presented. The mass flows and pressures, as they read in the interface from the 1D software, are plot over time for the different inlets and the outlet. They are superrimposed to the results that emerged from running only the 1D cooling system with the standard pump model, for both cases of opening or closing the thermostat. Mass flows are the same for the coupled system and the 1D, while pressures were adjusted to the new pressure drop that the 3D introduced after coupling. More specifically, for the period that the thermostat position is closed, the pressure at the inlets from the radiator and the tank is decreased compared to the 1D. However, the pressure at the inlet from thermostat and HVAC are increased. Smaller increase is shown for the pressure at the inlet from TOC, while the pressure at the outlet is the same as the 1D because it is set explicitly with the boundary. On the other hand, when the position of the thermostat is open there is still an increase in the pressure for the thermostat but it the offset is less as for the closed thermostat result. The pressure at the inlet from the radiator and the tank are still decreased but also with less percentage of offset in the value compared to the 1D. Flows from TOC and HVAC showed almost the same pressure as the 1D for this period. The simulations were concluded with the opposite configuration that the thermostat moved from open to closed position. The new points that the simulation settled were the same as for the previous configuration that is presented in Figure 4.12, which meant that the system behaved the same and this configuration did not require further investigation.



(a)



(b)

Figure 4.10: *Evolution of the a) pressure and b) velocity at the splitter bear the pump near the moment of changing the thermostat position (t=220s).*

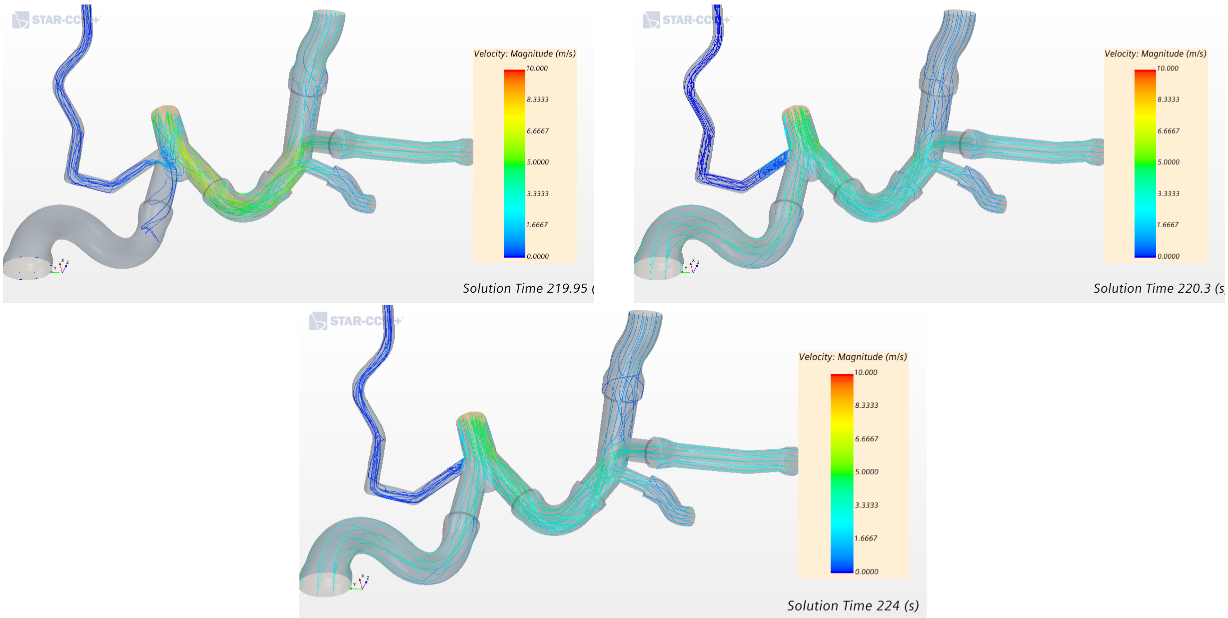


Figure 4.11: *Evolution of streamlines on the system near the moment of changing the thermostat position (t=220s).*

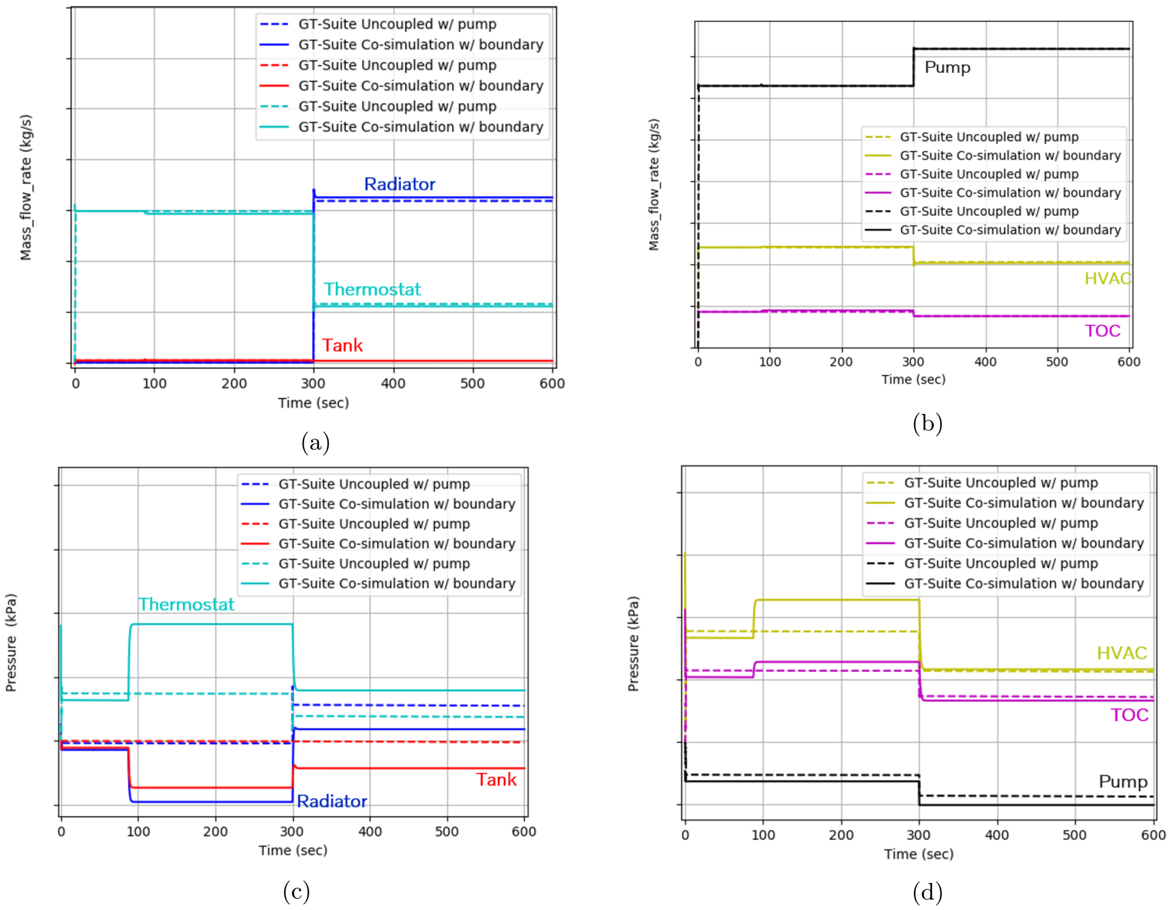


Figure 4.12: *a) Mass flows at inlet from radiator, expansion tank and thermostat HVAC ,b) at inlet from HVAC, TOC and outlet to pump. c) and d) Pressures for each interface of the system. Two-way coupling at t=82.5s and change of thermostat position at t=300s*

5 Discussion

In this project the objective was to establish a functioning co-simulation between 1D and 3D for the cooling system of a vehicle. After performing the first case and obtaining the results, the behaviour of the system was predictable. A slight variation in the pressure drop is obtained when a part of the system changes from being modeled in 1D to modeled in 3D. As the pump works on higher speed and the mass flow through the system increases, this offset would increase almost linearly. The settings of the configuration are a factor for achieving a smooth coupling point. The end result would be different only for cases that the 3D did not converge sufficiently between inner iterations. This effect is seen in the results in which the maximum inner iterations are too few or the continuity criteria is too big. Moreover, proper initialization is required otherwise the 1D would be introduced with very abrupt changes in the interfaces at the point of two-way coupling and could cause errors in the results. Initialization could be obtained by running a steady-state simulation, but also for some simulations it was sufficient to initialize the domain with a transient simulation that has a long one-way coupling period. For this period the boundaries are transferred from 1D to 3D only, forcing the 3D to perform an implicit unsteady simulation with constant boundaries.

In the first two cases slight variations were observed in the pressure drop of the 1D and the coupled 1D-3D simulations. Two different explanations could be given. The case of the straight pipe showed higher pressure drop for the co-simulation as seen in Figure 4.3, somethinng that can be caused by the flow behaviour in the entrance. More specifically, the entrance is given a uniform mass flow, which means for a constant density case a uniform velocity profile in the cross-section. The entrance region to an enclosed pipe is the region before the creation of the boundary layer. At this region the friction near the walls is much higher and for the flow to adjust to their existence hence the higher pressure drop. For the second case it was observed lower pressure drop for the co-simulation compared to the 1D model. Here the general believe is that it is possible for the 1D model for the hose with the bends to be created by the software with higher energy losses from the emperical model. The reason for this may be that is preferable to over-estimate the behaviour (pressure losses in this case) of a cooling system, rather than under-estimating. Under-estimating the losses in a cooling system can lead to unacceptable results from simulations. The higher the mass flow is, the higher the pressure drop for a flow though pipes for both cases. Of course these two explanations of the behaviour are just comments on the trend, but there would be more depth in the discussion if the results from the 3D simulation would be validated with experimental data. As mentioned in Chapter 1.3, this option was not offered in this project. It is a general belief that the 3D results are in most of the cases more precise than a 1D model, but more validation on the results of the co-simulation is suggested for further use. The turbulence models showed no significant variations between each other, making acceptable for each of them to be used for such hoses. Also when they were tested with more refined meshes for the two different models they showed almost the same result. This in effect, permitted the use of the model with the least computational cost, that was the k-epsilon model, and mesh refinement that was giving wall $y^+ \sim 10$.The maximum limit of the y^+ was 30 in order to use this model, as it was mentioned by the software company.

For the flow splitters at cases 3 and 4, different pump models were used in order the co-simulation to function. The flow splitter at the bypass was converged with the standard pump model since it was not directly before the pump. The coupling was smooth with almost identical mass flows between the co-simulation and the 1D simulation for all the three operating points. Variations were noticed in the pressure drops, as in cases 1 and 2. The flow from thermostat that in the 3D part, that comes from the top, had increased pressure drop. Flow that come from the HVAC and is the first connection that the flow from thermostat meets had also increased pressure drop. On the contrary, the flow from TOC that is the last flow coming to the splitter, and is located closest to to the outlet than the rest, had decreased pressure drop in the coupled 1D-3D system. If there are no inaccuracies in the results (which we can not be aware of, since there was not validation with experimantal data), it seems that the 3D can understand better the behaviour of a splitter concerning the distance of each inflow and outflow from the main mixing region of the flow. Other than this suggestion, the most important outcome of this case was that all the settings that had been selected, as a result of the previous simulations, functioned properly to this co-simulation and that a coupled system that is able to to simulate a small sub-component in 3D was able to perform very fast co-simulations. This could improve investigations that focus on small parts of the cooling system, but prefer the rest of the system to be modeled in 1D. There should be exceptions though, as it was proven from the case of the flow splitter before the pump.

For the case 4, the problem of using the standard pump model with the performance map was not feasible. More investigation could be performed to identify the problem, since investigations that were performed in this project to this problem were inconclusive of the cause of it. To give an outlook, investigations that could locate the problem in the expansion tank modeling in 1D or in turbulent and three-dimensional flow in the outlet of the splitter did not help to locate the cause. Especially, in the outlet that is also the inlet to the pump model in 1D, the pipe was elongated in order to achieve more stable and one-dimensional flow behaviour, but the oscillations that are reported in the Chapter 4.4 were still observed. As for the results with the constant boundary values in the pump, they showed the same values in the mass flows of the system and the pressure drop compared to the 1D model. The only significant change in the behaviour of the co-simulation was noted for the pressure in the inlet of the tank, which appears to be highly over-estimated by the 1D-system. In this case we can take into consideration that the flow in the inlet of the splitter from the expansion tank has low mass flow, with Reynolds number much smaller than the limit for turbulence. This part was nevertheless modeled with the same turbulent model as the rest of the system. Turbulence in low-Reynolds number is highly difficult to model and an important topic of research in the academia, but was out of scope in this project.

The last case of co-simulation introduced a much bigger part of the cooling system in 3D. The settings of the previous simulation for the 3D system were almost sufficient, with a minor tweek that was needed in the convergence criteria and under-relaxation factors to achieve faster convergence and stable solution. The value for the parameter wall $y^+ \sim 10$ was still applicable for convergence in 3D and not higher refinement was required. The introduction of the transient variation of the boundary conditions, the instanteneous change of thermostat position, was resolved smoothly by the system. The mass flows in the system remained same as the 1D simulation would provide with the standard pump model. For the result of the pressure, it can be compared the behaviour of the pressure drop for this co-simulation with the with the co-simulations for the hose and the bypass flow splitter (cases 2 and 3) at the same operating point at 4000 rpm. The pressure drop offset between the latter two cases was approximately 3 and 5 kPa, while the co-simulation of the bigger part when the thermostat was open (as was for the two previous cases) gave offset in pressure drops from a specific inlet of around 1 to 5 kPa for each inlet. The 3D representations of the flow and the pressure showed that high vorticity is present in the bend between the bypass splitter and the splitter before the pump. The pressure drops show that coupling didn't largely affect the pressures in the results. For the case when the thermostat was closed higher inaccuracy in the pressure estimation between the 1D system and the coupled 1D-3D system was observed, with the pressure in the inlet from the thermostat to be around 10 kPa increased, from the tank 7 kPa decreased and from the radiator 10 kPa decreased. Finally, when the case that the thermostat was initially open and afterwards closed, the co-simulation reacted the same way as the former configuration. The final results were the exactly the same while the reaction to the boundary variation was similarly smooth.

6 Conclusions and future work

6.1 Conclusions

The objective to establish a co-simulation for the cooling system with the coupling of 1D and 3D CFD software was successful. The option to perform a co-simulation with multiphase phenomena captured in 3D was not available because of limitations from STAR-CCM+ co-simulations availability. For the coupling of the software the settings could be a factor to achieve the proper final result.

The results focused on the pressure drop of the system and showed slight deviations between the coupled and the 1D simulation. It could be that the 3D was slightly more accurate in predicting the correct behaviour but the lack of experimental data does not validate this suggestion. The k-epsilon turbulence model was used after noticing similar behaviour with the k-omega, but with faster computation and less problem in converging.

The cases of a straight pipe, a curved hose and a flow splitter were simulated as the 3D part of the co-simulation, at different operating points, and the result showed no spikes or oscillations at the point of coupling. The case of a flow splitter located before the pump needed a different pump model in 1D to function but the results showed similar behaviour as for the previous cases. Finally, a bigger part of the cooling system was investigated in the 3D part of the co-simulation, adding an instantaneous change of boundary in 1D. The co-simulation reacted smoothly to the transient complexity that was introduced in the system, while the pressure drop showed similar trend with the rest cases that had slight offset to the single 1D simulations.

6.2 Future work and suggestions

Since the initial goal of including multiphase modeling in the 3D part of the coupled system was not achieved, it remains a topic to be investigated in the future. However, this project has introduced the main method for coupling the two softwares and it will accelerate that process. In order to perform a coupled 1D-3D simulation with multiphase flow in 3D, it would require the 3D software to upgrade its coupling capabilities for the multiphase solvers. Another option would be to use open source 3D software that are available. They also require investigation of their coupling capability, but the 1D software lists the open software CFD solver OpenFoam as an option for co-simulation. Moreover, the co-simulation is performed with an explicit coupling of the two software, while the 3D software controlling it. A proposal is to create a programming script that would have the ability to perform the controlling of the co-simulation and launch separately the two software. It would run both of them standalone for one time-step and then it would export their boundary data in files. At next these files should be read by the other software. After adjusting the boundaries the next time-step computation would be performed and the co-simulation would continue with this iterating process.

There can be introduced three cases that multiphase flow simulations could be performed in cooling systems. Firstly, boiling can be observed in problems that locally there is high heat flux from a wall section in the cooling system, which is called nucleate boiling. The method that can be suggested to model this is a segregated Euler-Euler solver, that will account for the mass fractions of vapor in the system, its movement downstream and its interaction with the main coolant flow. Evaporation model for these bubbles is also available, to identify their interaction with the flow and if and how they will dissolve. The 1D will be able to receive multiphase flow, but its modeling of multiphase flow interaction in the 1D will be limited, which means it will mostly transport it in the cooling pipes.

A second type of multiphase physics will be cavitation at low pressure regions. Vapor bubbles again will be created and travel downstream, but in this case modeling could not be achieved with an Euler-Euler solver. The volume of fluids method (VOF) offers the cavitation model in the STAR-CCM+, but such a model requires much higher refinement in the mesh of the 3D domain. Computations will increase dramatically. Cavitation models focus to estimate the creation of bubbles and the evolution of their radius.

Lastly in the cooling system there is existence of air. Most of the times it is located in the expansion tank but also air or vapor could be trapped in the radiator. To determine the 3D behaviour of the expansion tank interaction between coolant, air and vapor can be modeled in 3D. It is suggested to be modeled with a method that the phases interface is better estimated, like VOF, which as mentioned will need high refinement in the mesh.

Another proposal is to model more than sections in 3D that may be located in different places of the cooling system. An example would be to model the curved hose of case 2 and the splitter before pump of case 4 in 3D with the same 3D simulation file. Different regions and interfaces can be used for each of them. In this way both parts that are located before and after the radiator can be modeled in 3D, whereas the radiator's behaviour will be given by 1D modeling with experimental data. In such a system, if there are phenomena that are created locally and travel downstream, the interaction of these with the flow in a different component can be accounted for. Sometimes in a cooling system, it can be observed such a behaviour with high interaction of phenomena much downstream than where they are anticipated. Trapped air or vapor downstream from their generation could be one example of these cases.

References

[1] B. Chaouat. The state of the art of hybrid RANS/LES modeling for the simulation of turbulent flows. *Flow, turbulence and combustion* **99**.2 (2017), 279–327.

[2] L. Davidson. *Fluid mechanics, turbulent flow and turbulence modeling.* 2020. URL: `http://www.tfd.chalmers.se/~lada/postscript_files/solids-and-fluids_turbulent-flow_turbulence-modelling.pdf`.

[3] E. discoveries. *How engine cooling system works?* 2019 (accessed June 11, 2020). URL: `https://engineeringdiscoveries.com/2019/05/24/how-engine-cooling-system-works/`.

[4] G. Gamma Technologies LLC. *GT-SUITE - Flow Theory Manual.* 2019.

[5] G. Gamma Technologies LLC. *GT-SUITE - STAR-CCM+ Coupling Manual and Tutorials.* 2019.

[6] M. Marchionni et al. An appraisal of proportional integral control strategies for small scale waste heat to power conversion units based on Organic Rankine Cycles. *Energy* **163** (Aug. 2018). DOI: `10.1016/j.energy.2018.08.156`.

[7] U. G. Riegler and M. Bargende. Direct coupled 1D/3D-CFD-computation (GT-Power/Star-CD) of the flow in the switch-over intake system of an 8-cylinder SI engine with external exhaust gas recirculation. *SAE Transactions* (2002), 1554–1565.

[8] S. Siemens PLM Software. *Simcenter STAR-CCM+ User Guide Version 2019.3.* 2019.

[9] R. Van Basshuysen and F. Schäfer. *Internal combustion engine handbook-basics, components, systems and perspectives.* Vol. 345. 2004.

[10] A. Vdovin. Cooling performance simulations in GT-Suite (2010).

[11] N. Watanabe, M. Kubo, and N. Yomoda. *An 1D-3D integrating numerical simulation for engine cooling problem.* Tech. rep. SAE Technical Paper, 2006.

[12] F. M. White. *Fluid Mechanics, in SI Units.* McGraw-Hill, New York, 2016.