Investigation of CFD conjugate heat transfer simulation methods for engine components at SCANIA CV AB

Master Thesis

Luis Iñaki Martinez (901012-T557)
Investigation of CFD conjugate heat transfer simulation methods for engine components at SCANNIA CV AB

Master Thesis

Luis Iñaki Martinez (901012-T557)

Academic supervisor: Petter Ekman
IEI, Linköpings Universitet

Industrial supervisor: Eric Furbo
Scania CV AB, Södertälje

Examiner: Roland Gårdhagen
IEI, Linköpings Universitet
Abstract

The main objective of this Master Thesis project is the development of a new methodology to perform Computational Fluid Dynamics (CFD) conjugate heat transfer simulations for internal combustion engines, at the Fluid and Combustion Simulations Department (NMGD) at Scania CV AB, Södertalje, Sweden. This new method allows to overcome the drawbacks identified in the former methodology, providing the ability to use the more advanced polyhedral mesh type to generate good quality grids in complex geometries like water cooling jackets, and integrating all the different components of the engine cylinder in one unique multi-material mesh. In the method developed, these advantages can be used while optimizing the process to perform the simulations, and obtaining improved accuracy in the temperature field of engine components surrounding the water cooling jacket when compared to the experimental data from Scania CV AB tests rigs.

The present work exposes the limitations encountered within the former methodology and presents a theoretical background to explain the physics involved, describing the computational tools and procedures to solve these complex fluid and thermal problems in a practical and cost-effective way, by the use of CFD.

A mesh sensitivity analysis performed during this study reveals that a mesh with low y+ values, close to 1 in the water cooling jacket, is needed to obtain an accurate temperature distribution along the cylinder head, as well as to accurately identify boiling regions in the coolant domain.

Another advantage of the proposed methodology is that it provides new capabilities like the implementation of thermal contact resistance in periodical contact regions of the engine components, improving the accuracy of the results in terms of temperature profiles of parts like valves, seats and guides.

The results from this project are satisfactory, providing a reliable new methodology for multi-material thermal simulations, improving the efficiency of the work to be performed in the NMGD department, with a better use of the available engineering and computational resources, simplifying all the stages of multi-material projects, from the geometry preparation and meshing, to the post-processing tasks.
# Nomenclature

## Latin

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T$</td>
<td>Temperature</td>
<td>K, °C</td>
</tr>
<tr>
<td>$A$</td>
<td>Area</td>
<td>m²</td>
</tr>
<tr>
<td>$U$</td>
<td>Velocity magnitude</td>
<td>m s⁻¹</td>
</tr>
<tr>
<td>$u_i$</td>
<td>Velocity in the x, y or z direction</td>
<td>m s⁻¹</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
<td>Pa, bar</td>
</tr>
<tr>
<td>$i$</td>
<td>Specific Internal energy</td>
<td>m² s⁻²</td>
</tr>
<tr>
<td>$y^+$</td>
<td>Non-dimensional wall distance</td>
<td>-</td>
</tr>
<tr>
<td>$y$</td>
<td>Distance between the wall and first mesh cell in the grid</td>
<td>m</td>
</tr>
<tr>
<td>$u_t$</td>
<td>Friction velocity</td>
<td>m s⁻¹</td>
</tr>
<tr>
<td>$U^+$</td>
<td>Non-dimensional velocity</td>
<td>-</td>
</tr>
<tr>
<td>$c_p$</td>
<td>Specific heat Capacity</td>
<td>J kg⁻¹ K⁻¹</td>
</tr>
<tr>
<td>$k_t$</td>
<td>Turbulent kinetic energy</td>
<td>m² s⁻²</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
<td>-</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandtl number</td>
<td>-</td>
</tr>
<tr>
<td>$q$</td>
<td>Heat</td>
<td>W</td>
</tr>
<tr>
<td>$h$</td>
<td>Convective heat transfer coefficient</td>
<td>W m⁻² K⁻¹</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
<td>s</td>
</tr>
<tr>
<td>$L$</td>
<td>Length Scale</td>
<td>m</td>
</tr>
<tr>
<td>$f$</td>
<td>Elliptic relaxation function</td>
<td>s⁻¹</td>
</tr>
<tr>
<td>$d$</td>
<td>Diameter / Distance</td>
<td>m</td>
</tr>
</tbody>
</table>

## Greek

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\mu$</td>
<td>Dynamics viscosity</td>
<td>kg m⁻¹ s⁻¹</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
<td>m² s⁻¹</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>Eddy viscosity</td>
<td>kg m⁻¹ s⁻¹</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>kg m⁻³</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Thermal conductivity</td>
<td>W m⁻¹ K⁻¹</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Turbulence dissipation</td>
<td>m² s⁻³</td>
</tr>
<tr>
<td>$\zeta$</td>
<td>Velocity scale ratio</td>
<td>-</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Emmisivity</td>
<td>-</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Stefan-Boltzmann constant (5.669x10⁻⁸)</td>
<td>W m⁻² K⁻⁴</td>
</tr>
<tr>
<td>$\sigma_T$</td>
<td>Turbulent Prandtl number</td>
<td>-</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>von Karman’s constant (κ=0.4)</td>
<td>-</td>
</tr>
</tbody>
</table>
Abbreviations and Acronyms

<table>
<thead>
<tr>
<th>Letter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NMGD</td>
<td>Fluid and Combustion Simulations Department for engine development at Scania CV AB</td>
</tr>
<tr>
<td>AVL Fire</td>
<td>Computational fluid dynamics software developed by AVL</td>
</tr>
<tr>
<td>ANSA</td>
<td>Pre-processing tool. Software developed by BETA CAE Systems</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational fluid dynamics</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Average Navier-Stokes Simulations</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulations</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>WCJ</td>
<td>Water cooling jacket of the internal combustion engine</td>
</tr>
<tr>
<td>CAD</td>
<td>Computer-aided Design</td>
</tr>
<tr>
<td>CAE</td>
<td>Computer-aided Engineering</td>
</tr>
<tr>
<td>HTC</td>
<td>Heat transfer coefficient</td>
</tr>
</tbody>
</table>
Contents

Abstract ........................................................................................................................................... i
Nomenclature ................................................................................................................................... ii
Contents ........................................................................................................................................... iv

1. Introduction ................................................................................................................................... 1
   1.1 Problem Background .................................................................................................................. 1
   1.2 Literature study .......................................................................................................................... 2
   1.3 Aim ........................................................................................................................................... 2

2. Theoretical Framework ................................................................................................................. 3
   2.1 Reciprocating Internal combustion engines .............................................................................. 3
      2.1.1 Engine components and cooling system ............................................................................. 3
   2.2 Computational Fluid Dynamics ................................................................................................. 4
      2.2.1 CFD Methodology ............................................................................................................ 5
      2.2.2 Fluid Dynamics ................................................................................................................. 6
      2.2.3 Turbulence Modelling ....................................................................................................... 6
      2.2.4 $\zeta$-f Turbulence model .................................................................................................. 8
      2.2.5 Hybrid wall treatment ....................................................................................................... 9
   2.3 Heat Transfer ............................................................................................................................. 10
      2.3.1 Conduction ........................................................................................................................ 10
      2.3.2 Convection ........................................................................................................................ 11
      2.3.3 Radiation .......................................................................................................................... 12
      2.3.4 Heat Transfer and CFD ..................................................................................................... 12
   2.4 Single-Phase Boiling ............................................................................................................... 13
      2.4.1 Nucleate Boiling .............................................................................................................. 13
      2.4.2 Chen Boiling Model ......................................................................................................... 15
      2.4.3 Boiling Departure Lift-off Model ....................................................................................... 16
      2.4.4 Transition to film boiling .................................................................................................. 17
   2.5 Polyhedral Mesh ....................................................................................................................... 17

3. Method .......................................................................................................................................... 19
   3.1 Stage 1: Simplified SCANIA engine geometry study .............................................................. 22
      3.1.1 Pre-processing stage ......................................................................................................... 22
      3.1.2 Setup and Simulation ....................................................................................................... 24
   3.2 Stage 2: Complete SCANIA Geometry (1-cylinder) ............................................................... 28
      3.2.1 Pre-processing stage ......................................................................................................... 28
      3.2.2 Setup and simulation ...................................................................................................... 36
      3.2.3 Efficient cluster use Study ............................................................................................... 37
1. Introduction

The cooling system for an internal combustion engine plays a major role in order to achieve the desired behavior and performance of the engine. The function of this system is to cool down the engine high temperatures generated in the combustion chamber to preserve the mechanical integrity of the parts.

Heat transfer analysis between the cooling system and the rest of engine parts like cylinder block, cylinder head, valves, seats and guides, is a main field of study when designing an engine. This analysis helps to identify areas of high temperature, defines the amount of coolant and conditions needed to dissipate heat, and define the path of the cooling channels through the engine components in combination with the rest of the engine systems.

Computational Fluid Dynamics (CFD) is a powerful tool to handle heat transfer problems, especially in presence of complex flow conditions like complicated flow trajectories around engine components where convective heat transfer coefficients are difficult to accurately predict.

A new steady state thermal simulation methodology wants to be studied and implemented at Scania CV AB for thermal analysis of internal combustion engine components. This new simulation method involves using the latest multi component simulation strategy in the AVL Fire code, that allows the engineer to prepare the geometry, create a high quality polyhedral meshes, and generate interfaces between the multiple components or domains, either solid or fluid parts of the engine, in a more automated and efficient way than the method currently used at the company.

Due to confidentiality reasons within Scania CV AB, certain results in this report are given in normalized values according to reference results specified in each case, and scales in certain figures are hidden to avoid showing exact values computed for the studied parameters.

1.1 Problem Background

Using CFD methods to analyze thermal simulations in engine components involves several stages, from the pre-processing, setup, simulations and post-processing. In presence of highly complex geometries like the ones from internal combustion engines, with multi-material parts and fluid-solid interactions, the pre-processing stage consumes a dominant amount of resources, and due to its complexity can lead to errors or inaccuracies in the simulations.

Scania CV AB works continuously to develop internal combustion engines which can achieve low pollutant emissions and high efficiency. It is important to find out the thermal loads of the engine components early in the development process. This requires high accuracy simulation methods that can predict the temperature of components as well as the cooling performance of the engine cooling system.

Former methodology implemented at Scania CV AB for thermal simulations of engine components is called AVL Code Coupling Interface (ACCI), and several drawbacks have been identified for this method. Within the limitations of the former methodology there is the need of generating several meshes, one mesh per domain to be analyzed; the limitation of using hexahedral dominant meshes, having the risk of bad quality cells in complex regions of the geometry; the manual identification and connection of domain interfaces instead of being done in an automated way; the limitation of implementing thermal contact resistance values, other than zero, in the domain interfaces; and the need of several software licenses depending on the amount of domains involved (one license per domain), significantly impacting the cost of these simulations.
1.2 Literature study

During recent years, several studies and investigations aiming for the development and implementation of polyhedral meshes have been done, due to its advantages while performing CFD computations on them, like the reduction in simulation time and improved grid quality over all the cells reducing the risk of skewness and orthogonality problems when working with complex geometries. Examples of this studies are exposed by Milovan Peric and Stephen Ferguson in their work summarized in “The advantages of polyhedral meshes” [1] within the computer software company CD-Adapco, which is used as a reference to this project.

Balafas [2] in his Master Thesis “Polyhedral Mesh generation for CFD-Analysis of complex structures” also present a methodology for the generation of polyhedral meshes and performs a comparison study against other type of meshes like hexahedral and tetrahedral arrangements, concluding that polyhedral meshes have an advantage against tetrahedral ones in terms of the amount of cells required to achieve the same level of accuracy, as well as in the reduction of simulation time. These are similar conclusions to previous studies on the field.

Regarding the modelling of turbulent flows, as in the case of this project for the simulation of coolant inside the engine water coolant jacket, AVL Fire support recommendation is the use of the $\zeta$-$f$ turbulence model developed by Hanjalic, Popovac and Hadziabdic in 2004. In their work titled “A robust near wall elliptic-relaxation eddy-viscosity turbulence model for CFD” [3] they present the $\zeta$-$f$ turbulence model and its advantages against the $k$-$\varepsilon$ turbulence model.

Another important feature regarding thermal analysis in internal combustion engines is the presence of boiling in the water cooling jackets. This can be studied by using a boiling module in the CFD thermal simulations. In 2004, Steiner developed a new boiling model to account for nucleate boiling formation in flows in contact with hot surfaces, this model is called the Boiling Departure lift-off (BDL) model, and it is presented and described in the articles “A wall heat transfer model for subcooled boiling flow” [4] and “Increased cooling power with nucleate boiling flow in automotive engine applications” [5]. Formation of nucleate boiling in the water cooling jacket of engines increment significantly the heat transfer from engine components to the coolant, increasing the efficiency of the engine cooling system.

1.3 Aim

The aim of the master thesis project is to investigate, evaluate and propose a new simulation methodology to perform multi-component simulations in one go, in order to reduce the lead time for such a simulation, optimize the use of software and engineering resources, reduce the error sources and get results comparable or better compared to the former methodology used at Scania CV AB and to experimental tests.

This new multi-material methodology, should be able to generate a discretized polyhedral mesh of the entire model with several parts or domains at once, identifying different components interfaces and connecting them properly, testing the advantages of the new software version of AVL Fire M available at Scania CV AB.

Important features of the new methodology that will be tested are: the generation of a polyhedral type mesh which provides good mesh quality for highly complex geometries simplifying the meshing refinement process; the complexity of implementing the multi-material approach within the AVL Fire code; the stability of the simulations when the boiling module is activated; and the possibility to add thermal contact resistance values for the simulation of periodic contact regions for different components like valves and seats (opening and closing of valves), while running steady state simulations.

Additional points to address during the project involves the successful preparation of the Computer Aided Design (CAD) models of the components, and how to create high quality geometries and meshes; to determine what is the level of refinement needed; how to run the simulations with an efficient use of the resources like computer cores, simulation time and software licenses in the cluster; and how to post-process the results in and efficient way.
2. Theoretical Framework

2.1 Reciprocating Internal combustion engines

Reciprocating internal combustion engines are piston engines, in which energy is transferred from a fluid (a gas or liquid) to a moving displacer or piston. They are part of the category of fluid energy machines, where mechanical energy is released in the form of useful work at the piston or at the crank mechanism [6].

In these machines, chemical energy is converted into mechanical energy as a result of the combustion of an ignitable mixture of air and fuel. In engines featuring internal combustion, the working fluid (air) is simultaneously the source of the oxygen necessary for combustion. Combustion of the fuel produces waste gas, which must be replaced as a gas exchange cycle prior to every working cycle, therefore this process is cyclical, differentiation being made between gasoline, diesel, depending on the combustion process [6].

2.1.1 Engine components and cooling system

Relevant combustion engine components in this study are presented in this section and correspond to the engine piston, the engine block, the liner, the cylinder head, the valves, the seats, and the guides.

The functions carried out by the piston include accepting the pressures created by the ignition of the fuel and air mixture, transferring these forces via the wristpin and the connecting rod to the crankshaft [6].

The engine block is the component that encloses the cylinders, the cooling jacket and the engine block shell.

The piston group is mounted in the cylinders or liners, which also supports the slip and sealing functions. They contribute to heat dissipation via the engine block or directly into the coolant. Most of today’s automotive engines are water cooled, the cylinders are surrounded by a water-filled cavity, a section of the water cooling jacket [6].

The cylinder head determines operating properties such as performance level, torque, exhaust emissions, fuel consumption, and acoustic properties. The cylinder head is the part of the engine that defines the arrangement of valves, as well as the design of the intake and exhaust ports to the combustion chamber. It also plays an important role in the cooling systems since the water cooling jacket passes through it, along complex channels, dissipating heat from all the components [6]. The main objective in cooling is create uniform cooling conditions for all cylinder segments. By simulation of the coolant flow, problematic areas such as the webs between the exhaust ports, can be engineered for complete reliability [6].

Intake and exhaust valves are precision engine components used to block gas flow ports and to control the exchange of gases in internal combustion engines, their function is to seal the working space inside the cylinder against the manifolds [6].

Intake valves are not subjected to extreme thermal loading and are cooled by the incoming gases and by thermal transmission at the seat and guides. On the other hand, exhaust valves are exposed to severe thermal loads and chemical corrosion. The different types of valves are manufactured using different materials matched to the functions they perform [6].

The seats for the exhaust valves are heavily impacted by heat and corrosion, so it is hard-faced with special treatment, this can reduce wear and enhance the sealing effect.

The guides ensure that the valves centers in the seats and that heat can be dissipated from the valve head, through the valve stern and to the cylinder head [6].
Due to the high temperatures generated in the combustion process, the engine needs to be cooled to protect its components and the lubricating oil. It is necessary to differentiate between direct and indirect engine cooling [6].

Direct cooling is accomplished by using air, either with or without the assistance of a fan. In the case of indirect cooling, the engine is cooled with a mixture of water, antifreeze, and corrosion inhibitors, or with oil (liquid cooling). In order to remove the heat to the environment a heat exchanger arrangement is used [6].

Peak temperatures of over 2000 °C occur inside the cylinder of an internal combustion engine, however, charge cycles and expansion processes between the ignitions result in far lower mean temperatures. Nevertheless, thermal overloading of the components exposed to the gas must be prevented and the lubricating properties of the oil film between piston and cylinder surfaces has to be maintained by cooling.

In water-cooled internal combustion engines, one third of the admitted fuel energy is discharged via the cooling system, a further third is lost via the exhaust gas, and one third is transformed into useful work depending on the combustion process.

2.2 Computational Fluid Dynamics

CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena by means of computer-based simulation. It consists of solving the discretized governing equations of fluid flow that describes the conservation laws of physics [7].

Governing equations of the flow for a Newtonian Fluid, a fluid in which the viscous stresses arising from its flow are linearly proportional to the local strain rate, are shown, where Eq. 1 corresponds to the conservation of mass; Eq. 2 corresponds to the momentum equation where the rate of change of momentum is equal to the sum of the forces on a fluid particle (Newton’s second law); and Eq. 3 corresponds to the energy equation, where the rate of energy is equal to the sum of the rate of heat addition and the rate of work done on a fluid particle (first law of thermodynamics) [7].

\[
\frac{\partial \rho}{\partial t} + \text{div}(\rho \vec{u}) = 0 \quad \text{(Eq.1)}
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \text{div}(\rho u_i \vec{u}) = -\frac{\partial p}{\partial x_i} + \text{div}(\mu \text{grad} u_i) + S_{M_i} \quad \text{(Eq.2)}
\]

\[
\frac{\partial (\rho i)}{\partial t} + \text{div}(\rho i \vec{u}) = -p \text{div} \vec{u} + \text{div}(\lambda \text{grad} T) + \Phi + S_i \quad \text{(Eq.3)}
\]

Where \( \rho \) is the density of the fluid; \( t \) is the time; \( \vec{u} \) is the fluid particle velocity vector; \( u_i \) is the velocity of the fluid particle in the corresponding \( x, y \) and \( z \)-direction; \( \mu \) is the dynamic viscosity; \( i \) is the internal energy; \( \lambda \) is the heat conduction coefficient; \( S \) corresponds to the sources terms; and \( \Phi \) are the dissipation terms.

With the continuity equation (Eq.1), the momentum Navier-Stokes Equations (Eq.2, Eq.3, Eq.4) and the Energy equation (Eq.5), together with the equations of state of the fluid, it is possible to solve non-turbulent flow problems.

The energy equation (Eq.5) is of interest to consider thermal and compression effects in the simulations. It is relevant to mention that for incompressible fluids, without density variations, the temperature is decoupled from the rest of the equations and only needs to be solved alongside the others if the problem involves heat transfer [7].
2.2.1 CFD Methodology

The methodology to solve CFD problems consists of three main steps: pre-processing, processing and simulations, and post-processing of the results.

During the pre-processing stage the procedure carried out involves the definitions of the geometry or regions of interest, a grid generation where the flow domains are split into smaller subdomains so the governing equations can be discretized and solved inside each of these subdomains, the selection of physical and chemical phenomena to model, the definition of the fluid properties, and the specification of the of appropriate boundary conditions of boundary cells to define the limits of the problem with the proper physical conditions for the model [7].

The most used simulation technique for this type of problems is the finite volume method, and it is implemented in most of the codes, like the AVL Fire code.

The finite volume method allows the representation and evaluation of partial differential equations in the form of algebraic equations, where the solution to a flow problem like velocity, pressure or temperature, is defined at nodes inside each cell of the discretized domains (cell center based), where the conservation laws are solved [7].

The AVL Fire CFD Solver employs the finite volume discretization method where general convex polyhedron type of cell can be used as shown in Figure 1 [8].

![Control volume for a polyhedral cell in the cell-based finite volume method. Reproduced from [8]](image)

The first step in the finite volume method is to divide the domain into discrete control volumes, in this case, the boundaries of each cell are placed between adjacent nodes [7].

The second step of the finite volume method is the discretization, which is the integration of the governing equations over the control volumes, having a clear physical interpretation since the flux entering a given volume is the same to the one leaving the adjacent volume. Discretized equations are set up at each of the nodal points in the mesh [7]. To calculate gradients and fluxes at the control volume faces an approximate distribution of properties between nodal points is used, if this approximation is linear is called the central differencing scheme, more complex approximations are also used depending of the complexity of the problem, like the Upwind or QUICK schemes [7].
The last step corresponds to the solution of the equations, the resulting system of linear algebraic equations is solved with any suitable matrix solution technique (e.g. Gauss-Seidel) [7].

The post-processing stage is where the results can be studied and analyzed by using versatile data visualization tools usually included in the software used [7].

2.2.2 Fluid Dynamics

Flows that can be found in engineering problems, like pipes or on flat plate boundary layers, as well as complicated three-dimensional ones like the coolant inside the water cooling jacket, become unstable above a certain Reynolds number [7].

The Reynolds number is a non-dimensional value, known as the ratio between inertial forces to viscous forces in a fluid, it is defined for a flow through a pipe as shown in Eq. 4 [7].

\[
Re = \frac{\rho u D}{\mu}
\] 

(Eq.4)

where \( \rho \) is the density of the fluid, \( u \) is the mean velocity of the fluid inside the pipe, \( D \) is the inner diameter of the pipe and \( \mu \) is the dynamic viscosity of the fluid.

At low Reynolds numbers flows are observed to become laminar, where the flow is smooth, predictable and present an orderly fashion. Meanwhile, for high Reynolds numbers flows become turbulent, characterized by a chaotic and random state of motion where the velocity and pressure change continuously with time [7]. For flows inside pipes, a low Reynolds number is considered to be below the Reynolds critical value of 2000, providing a laminar flow, while for high Reynolds number above 2000 the flow becomes turbulent. For other types of flow inside different geometries the critical Reynolds number can be different.

Due to the turbulence effects and its effective mixing of the flow, heat mass and momentum are effectively exchanged, which is usually a desired behavior for heat transfer problems. Turbulence causes in the flow the formation of eddies, which are identified by the swirling of the fluid and reverse currents created in the turbulent regime [7]. These eddies have a wide range of length and time scales that interact dynamically in complex ways. Due to the importance of the presence of turbulence in many engineering applications, important effort has been done in research and development of numerical method to capture the relevant effects related to it [7].

2.2.3 Turbulence Modelling

There are three categories for methods studying turbulent flows, the first one corresponds to the turbulence models for Reynolds-averaged Navier-Stokes (RANS) equations, where attention is aimed to the mean flow and the effects on turbulence in the mean flow properties. In this category, the Navier-Stokes equations are time averaged and the extra terms that appear in the Reynolds-averaged equations are modelled with turbulence models like the \( k-\varepsilon \) model. The computational resources required for accurate flow computations are modest, making this approach the main one used in engineering flow calculations over the last decades [7].

The other two categories correspond to Large eddy simulations (LES), and Direct numerical simulations (DNS). For LES methods, large eddies, together with the mean flow, are computed and the smaller ones are modelled, they demand more computational resources but are also being used in industry in CFD problems with complex geometries [7].

DNS simulations compute the mean flow and all turbulent velocity fluctuations. These calculations are highly costly in terms on computing resources, so this approach is not used in industrial applications [7].
The RANS equations can be obtained by applying the Reynolds decomposition technique to the Navier-Stokes equations. The Reynolds decompositions refers to the separation a flow variable into a mean (time-averaged) component and a fluctuating component, for example u-velocity (Eq.5):

\[ u(x, t) = \bar{u}(x) + u'(x, t) \]  
(Eq.5)

In Eq. 7, \( u(x, t) \) represent the velocity in the x-direction, \( \bar{u}(x) \) is the mean component and \( u'(x, t) \) is the fluctuating component.

By substituting the flow variables with the new components in the original equations and taking time average, the RANS equations are obtained. It is important to mentions that during this process six new terms (unknowns) appear in the time-average momentum equations, this terms are called the Reynolds stresses:

\[ -\rho \bar{u}'^2, -\rho \bar{v}'^2, -\rho \bar{w}'^2, -\rho u'v', -\rho u'w', -\rho v'w' \]  
(three normal stresses and three shear stresses) [7]. Similarly, time-average transport equations will have extra terms. The RANS equations for incompressible flows are shown, and correspond to Eq.6 and Eq.7 [7]:

\[ \text{div}(\bar{u}) = 0 \]  
(Eq.6)

\[ \frac{\rho \partial (u_i)}{\partial t} + \rho \text{div}(u_i \bar{u}) = - \frac{\partial p}{\partial x_i} + \mu \text{div(grad } u_i) + \left[ \frac{\partial(-\rho u_i' u_j')}{\partial x_j} \right] \]  
(Eq.7)

After this is done, the closure problem in computational fluid dynamics have been introduced, since there are more unknowns than equations. To deal with the closure problem, a Bousinesq approach can be applied.

Boussinesq proposed in 1877 that Reynolds stresses might be proportional to mean rates of deformation as shown in Eq. 8:

\[ -\rho u'_i u'_j = \mu_r \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) + \frac{2}{3} \rho k_t \delta_{ij} \]  
(Eq.8)

where \( \mu_r \) is the turbulent or eddy viscosity, \( k_t \) is the turbulent kinetic energy per unit mass and \( \delta_{ij} \) is the Kronecker delta.

Around Boussinesq hypothesis several turbulence models have been developed, being the most used and validated the k-\( \varepsilon \) model.

The k-\( \varepsilon \) model is the most used turbulence model, particularly for industrial problems and has been implemented into most CFD codes. It is numerically robust and has been tested in a broad variety of flows, including heat transfer, combustion, free surface and two-phase flow [8].

However, during the years, there have been found limitations or deficiencies of the k-\( \varepsilon \) turbulence model, having poor performance in flows near the walls, in prediction of separation (adverse pressure gradient) and with anisotropic turbulence [3].

For this reason, new RANS turbulence models have been developed to achieve a better representation of the flow in comparison with experiments and expensive simulations like LES or DNS. One of these advanced RANS models, implemented in the AVL Fire code, corresponds to the \( \xi-f \) model.
2.2.4 ζ-f Turbulence model

The ζ-f model was developed by Hanjalic, Popovac and Hadziabdic (2004). This model corresponds to a modification of the \( \bar{v}^2 \)-f model in order to improve its numerical stability.

The \( \bar{v}^2 \)-f model is developed by Durbin (1991) [9], it introduces an additional (wall-normal) velocity scale \( \bar{v}^2 \) and an elliptic relaxation concept to sensitize \( \bar{v}^2 \) to the inviscid wall blocking effect, the model does not use the conventional practice of introducing empirical damping functions [3].

The \( \bar{v}^2 \)-f model is a much better option than the conventional near-wall \( k-\varepsilon \) and similar models when considering complex three-dimensional flows, with strong secondary circulation, rotation and swirl, where the evolution of the complete stress field may be essential for proper reproduction of flow features, although it is still inferior to second-moment and advance non-linear eddy viscosity models. The drawback with the \( \bar{v}^2 \)-f model is that it has features that affects its computational efficiency and stability [3].

The ζ-f model aims to improve the numerical stability of the original \( \bar{v}^2 \)-f by solving a transport equation for the velocity scale ratio \( \zeta = \bar{v}^2 / k_t \) instead of the velocity scale \( \bar{v}^2 \). This model has demonstrated to be a good compromise between the model simplicity and its performance in capturing near-wall phenomena in complex flows [10].

The set of equations for the ζ-f model correspond to Eq. 9 to Eq. 13 [10]:

\[
\mu_{\tau} = C_{\mu} \zeta k_t \tau \quad \text{(Eq.9)}
\]

\[
\frac{D\zeta}{Dt} = f - \zeta \frac{k_{\zeta}}{k_t} P_{k_t} + \frac{\partial}{\partial x_k} \left[ \left( \frac{\mu_{\tau}}{\sigma_{\zeta}} \right) \frac{\partial \zeta}{\partial x_k} \right] \quad \text{(Eq.10)}
\]

\[
L^2 \nabla^2 f - f = \frac{1}{\tau} \left( C_1 - 1 + C'_2 \frac{P_{k_t}}{\epsilon} \right) \left( \zeta - \frac{2}{3} \right) \quad \text{(Eq.11)}
\]

\[
\frac{Dk_t}{Dt} = P_{k_t} - \epsilon + \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu_{\tau}}{\sigma_{k_t}} \right) \frac{\partial k_t}{\partial x_j} \right] \quad \text{(Eq.12)}
\]

\[
\frac{D\epsilon}{Dt} = \frac{(C_{\varepsilon_1} P_{k} - C_{\varepsilon_2} \epsilon)}{\tau} + \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu_{\tau}}{\sigma_{\epsilon}} \right) \frac{\partial \epsilon}{\partial x_j} \right] \quad \text{(Eq.13)}
\]

Where ζ is the velocity scale ratio and \( f \) is the elliptic relaxation function. The time (\( \tau \)) and length scale (\( L \)) are limited with the Kolmogorov scales as the lower bounds, and Durbin’s realizability constraints providing the upper bounds, as shown in Eq. 14 and Eq. 15:

\[
\tau = \max \left[ \min \left( \frac{k_t}{\epsilon}, \sqrt{6} C_{\mu} |S| \right), C_{\tau} \left( \frac{\mu^3}{\varepsilon} \right)^{1/2} \right] \quad \text{(Eq.14)}
\]

\[
L = C_L \max \left[ \min \left( \frac{k_t^{3/2}}{\epsilon}, \frac{k_t^{1/2}}{\sqrt{6} C_{\mu} |S| \zeta} \right), C_{\eta} \left( \frac{\mu^3}{\varepsilon} \right)^{1/4} \right] \quad \text{(Eq.15)}
\]
The $\zeta$-$f$ model coefficients are shown in Table 1.

**Tab. 1: Coefficients used for the $\zeta$-$f$ model turbulence model.**

<table>
<thead>
<tr>
<th>$C_{m}^{\zeta}$</th>
<th>$C_{e1}$</th>
<th>$C_{e2}$</th>
<th>$C_{1}$</th>
<th>$C_{2}'$</th>
<th>$\sigma_k$</th>
<th>$\sigma_{\varepsilon}$</th>
<th>$\sigma_{\zeta}$</th>
<th>$C_{\tau}$</th>
<th>$C_{L}$</th>
<th>$C_{n}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.22</td>
<td>1.4+($1+0.012/\zeta$)</td>
<td>1.9</td>
<td>1.4</td>
<td>0.65</td>
<td>1</td>
<td>1.3</td>
<td>1.2</td>
<td>6.0</td>
<td>0.36</td>
<td>85</td>
</tr>
</tbody>
</table>

### 2.2.5 Hybrid wall treatment

In fluid flows against no slip walls there is the development of a boundary layer characterized by three different regions, the viscous sub-layer where viscous effects are dominant against the inertial effects, the buffer region which is known as a transition region to the turbulent flow, and the fully developed turbulent region where inertial effects are dominant over viscous effects in the fluid [7]. The $y^+$ value is the non-dimensional wall distance shown in Eq. 16, and is used as a measure of where the first cell in the grid is placed inside the boundary layer.

$$y^+ = \frac{u_t y}{v} \quad \text{(Eq.16)}$$

where $u_t$ is the friction velocity at the nearest wall, $y$ is the distance between the wall and the first mesh cell in the grid, and $v$ is the kinematic viscosity of the fluid.

Depending on the values of $y^+$ obtained in the generated mesh for a CFD simulation, it will be possible to capture, or not, the physics happening inside the boundary layer. If $y^+$ values are high, for example higher than 5, then it is common to introduce wall functions to model the boundary layer effects.

A hybrid wall treatment is implemented in the AVL Fire code, this ensures a gradual change between viscous sublayer formulations and the wall functions. The hybrid wall treatment was developed by Popovac and Hanjalic in 2006, who extended Kader’s (1981) initial proposal for the description of temperature profile in the wall boundary layer, and of all turbulence properties [8].

The hybrid wall treatment reduces either to the integration to the wall when the first near-wall cell is in the viscous sub-layer ($y^+<5$), or to the appropriate wall function when it lies in the turbulent region ($30<y^+<500$). When the first grid node is in the buffer region ($5<y^+<30$), the boundary conditions are provided from blending the viscous and fully turbulent limits using exponential blending functions. This blending is based on a generalization of the expressions for the mean velocity and temperature profiles of Kader (1981) that approximate reasonably well the whole wall region of a boundary layer, including its viscous-conductive and turbulent logarithmic layers [10].

The hybrid wall treatment can be applied in conjunction with any turbulence model that permits integration to the wall and with any wall functions. However, it is advisable to use a well-tuned, physically well-justified and robust integration to the wall model, preferably without empirical damping functions, which has been proved to successfully reproduce properties of a minimum set of generic flows exhibiting various non-equilibrium effects (strong pressure gradients, separation, impingement, and others). The best suited turbulence model for the hybrid wall treatment, recommended by the authors of the method, is the $\zeta$-$f$ model [10].

The hybrid wall treatment is implemented by Eq.17, Eq.18 and Eq.19 [8]:

$$U^+ = y^+ e^{-\gamma} + \frac{1}{\kappa} \ln(Ey^+) e^{-\gamma} \quad \text{(Eq.17)}$$
\[ \mu_w = \mu \frac{y^+}{U'^+} \quad \text{(Eq.18)} \]

\[ \Gamma = \frac{0.01(Pr y^+)^4}{1 + 5Pr^3 y^+} \quad \text{(Eq.19)} \]

Where \( U^+ \) is the non-dimensional velocity defined in the law of the wall [7], \( y^+ \) is the non-dimensional wall distance (viscous length), \( \kappa \) is the von Karman’s constant (\( \kappa = 0.4 \)) and \( E \) is an additive constant (\( E = 9.8 \)). The subscript “P” denotes the wall nearest cell-center.

### 2.3 Heat Transfer

Heat transfer is the science that aims to predict the energy transfer between material bodies or fluids because of a temperature difference, this energy transfer is defined as heat. Heat transfer not only explains how energy is be transferred, but predicts the rate at which the exchange will take place under certain specified conditions. Heat transfer complements the first and second laws of thermodynamics by adding additional experimental rules used to establish energy-transfer rates [11].

There are three modes of heat transfer, depending on the characteristics and the medium at which the heat exchange takes place, it can be defined as conduction, convection or radiation.

#### 2.3.1 Conduction

Conduction is referred to the energy transfer in a body, where a temperature gradient exists, from the high-temperature region to the low-temperature region. The defining equation for thermal conductivity is called the Fourier’s law of heat conduction where \( \lambda \) is the heat conduction coefficient [W/m K], \( q \) is heat transfer rate [W] and \( \frac{\partial T}{\partial x} \) is the temperature gradient in the direction of the heat flow [\( \Delta K/m \)], as shown in Eq.20:

\[ q = -\lambda A \frac{\partial T}{\partial x} \quad \text{(Eq.20)} \]

The thermal conductivity, \( \lambda \), is a property of the material (solid or fluid) where the heat transfer phenomena occurs, it is the ability of a material to conduct heat, \( A \) is the surface area and \( T \) corresponds to the temperature. In general, thermal conductivity is strongly temperature dependent. The one-dimensional heat conduction equation (Eq.21) is shown [11]:

\[ \frac{\partial}{\partial x} \left( \lambda \frac{\partial T}{\partial x} \right) + \dot{q} = \rho c_p \frac{\partial T}{\partial t} \quad \text{(Eq.21)} \]

where \( \dot{q} \) corresponds to the energy generated per unit volume [W/m³]; \( c_p \) is the specific heat of the material [J/kg K]; and \( \rho \) is the density [kg/m³]

Similarly, this can be applied to treat more than one dimensional heat flow adding the remaining coordinate directions. If the problem can be considered to be stationary, like the study cases in this project, the time dependent terms can be removed from the equations.

Thermal contact resistance values can be used in interfaces between different components or materials in cases where the conduction process in not continuous. Thermal contact
resistance is defined as the inverse of the thermal conductance coefficient and values are usually specified in units of \([m^2\cdot K/W]\).

### 2.3.2 Convection

Convection heat transfer takes place usually between a solid wall and a fluid, where the temperature gradient at the wall depends on the flow field of the fluid against it.

To express the overall effect of convection, we use Newton’s law of cooling (Eq. 22) [11]:

\[
q = hA(T_w - T_\infty)
\]

(Eq. 22)

The quantity \(h\) [W/m\(^2\)K] is called the convection heat transfer coefficient, \(T_w\) is the temperature of the wall and \(T_\infty\) the temperature of the fluid free stream (outside the thermal boundary layer).

An analytical calculation of \(h\) can be done for simple systems, although for complex situations it must be determined experimentally because of the complexity of the flow field. As mentioned before, the convective heat transfer coefficient is highly dependent on the flow characteristics, especially on the velocity of the fluid. When velocity of the fluid is caused by external sources this can be defined as forced convection. On the other hand, when there is no external source of motion to the fluid, and the movement of it is a result of density gradients next to the wall, this is called natural or free convection [11].

Another type of convection is the one known as boiling convection, characterized by an important increment in the heat flux when nucleate boiling takes place on the fluid. Boiling convection is an important phenomenon present in many industrial applications like in the cooling system of an internal combustion engine, and it is discussed in more details in Section 2.4.

For convective heat transfer occurring in fluid internal boundaries and fluid-solid interfaces, on CFD problems, the conjugate heat transfer approach is used. In this case, these regions are treated as boundary conditions [8].

The enthalpy (energy) near wall treatment is applied by defining the near wall thermal conductivity as shown in Eq. 23 [8]:

\[
\lambda_w = \begin{cases} 
\lambda & \text{if } y_p^+ < y_T^+ \\
\frac{y_p^+ \mu c_p}{U_p^+ \kappa} \frac{\sigma_T}{\sigma_f} & \text{if } y_p^+ \geq y_T^+
\end{cases}
\]

(Eq. 23)

where the viscous sub-layer resistance factor \(Y\) is given by Eq. 24:

\[
Y = 9.24 \left[ \left( \frac{Pr}{\sigma_T} \right)^{0.75} - 1 \right] \left[ 1 + 0.28 \exp \left( -0.007 \frac{Pr}{\sigma_T} \right) \right]
\]

(Eq. 24)

the thermal boundary layer thickness \(y_T^+\) is obtained from the intersection of the linear and logarithmic law for temperature as shown in Eq. 25:

\[
\frac{Pr}{\sigma_T} y_T^+ = \frac{1}{\kappa} \ln (E_T y_T^+) + Y , \ E_T \approx 9.8
\]

(Eq. 25)

c\(_p\) corresponds to the specific heat capacity of the fluid, \(\sigma_T\) is the turbulent Prandtl number, \(\kappa\) is the von Karman’s constant (\(\kappa = 0.4\)) and \(U_p^+\) the non-dimensional velocity, \(y_p^+\) is the non-dimensional wall distance defined by Eq. 26 [8]:

\[
y_p^+ = \frac{\rho p c_p \mu^{1/4} K_p^{1/2} \Delta n_p}{\mu}
\]

(Eq. 26)
where $C_\mu$ is a constant ($C_\mu = 0.22$), $K_P$ the kinetic energy in the near wall node “P”, and $\Delta n_P$ the normal distance from near wall node “P” to the wall.

From the shown equations, it is possible to define the boundary heat flux $q_b$ (Eq. 27), the internal heat transfer coefficient $h_i$ (Eq. 28) and the internal cell temperature normal to the wall $\bar{T}_P$ (Eq. 29)

$$q_b = h_i(T_b - \bar{T}_P) \quad \text{(Eq. 27)}$$

$$h_i = \lambda_w \frac{A_b}{A_b \cdot d_b} \quad \text{(Eq. 28)}$$

$$\bar{T}_P = T_P + \nabla T_P \cdot \left( \frac{d_b - \frac{A_b \cdot d_b}{A_b^2}}{A_b^2} \right) \quad \text{(Eq. 29)}$$

$T_b$ corresponds to the temperature at the boundary face, $A_b$ is the area of the boundary face, $d_b$ is the distance from the cell center to the boundary face and $T_P$ the internal cell temperature.

Figure 1 shows the faces and dimensions used in the previous equations for a polyhedral cell. [8]

### 2.3.3 Radiation

Thermal radiation is a mechanism where no medium is needed for heat to be transferred, like in the case for conduction and convection. The mechanism in this case is electromagnetic radiation which is propagated as a result of a temperature difference.

Radiation phenomena can be complex and the calculations are usually simplified as shown in Eq.30 [11]:

$$q = \epsilon \sigma A(T_1^4 - T_2^4) \quad \text{(Eq. 30)}$$

Where $\epsilon$ is the emissivity of the surfaces, $\sigma$ corresponds to the Stefan-Boltzmann constant $5.669 \times 10^{-8}$ [W/m$^2$K$^4$], $T_1$ is the surface 1 temperature [K], and $T_2$ is the surface 2 temperature [K].

Radiation heat transfer can represent up to 20% of the total for diesel engines inside the combustion chamber, affecting the temperatures of the liner walls, being an important factor to include in combustion analysis [12] [13]. However, radiation plays a minor role within engine cooling systems, where the most important or dominant heat transfer mechanisms are convection in the coolant, and conduction through the solid parts of the engine. For this reason, radiation effects are generally neglected when doing CFD thermal analysis of engine components and cooling systems [12] [13].

### 2.3.4 Heat Transfer and CFD

Heat transfer can be analyzed or studied by the use of Computational Fluid Dynamics (CFD). As it was mentioned in the previous chapter, the energy equation for fluid dynamics problems is the one that involves the temperature variable, together with the thermal conductivity property.

In cases when liquids and gases flow at slow speed they behave as incompressible fluids. With no variation in density there is no linkage between the energy equation and the mass...
conservation and momentum equations. In this case, the flow field can be solved by considering the mass conservation and momentum equations only [7].

Since this problem involves heat transfer, the energy equation is solved alongside the others to resolve the temperature field.

For the case of the solid components, there is no need to solve the fluid transport equations, but to be able to obtain the temperature distribution along them it is needed to solve the energy equation on these domains.

In the interfaces between fluid and solid domains convection takes place as the predominant form of heat transfer, and these regions are solved by CFD using the conjugate heat transfer approach explained in the Convection section of the present report. The AVL Fire code is capable to model both solid and fluid domains and the interfaces between them, using the conjugate heat transfer method, to obtain the temperature distribution and heat transfer coefficients. The AVL Fire solver uses the enthalpy (energy) near-wall treatment by defining a near-wall thermal conductivity ($\lambda_w$). This is done to model the convective phenomena occurring inside the thermal boundary layer [8] (See Section 2.3.2).

### 2.4 Single-Phase Boiling

#### 2.4.1 Nucleate Boiling

Boiling may occur when a surface is exposed to a liquid and is maintained at a temperature above the saturation temperature of the liquid, and the heat flux will depend on the difference in the temperature between the surface and the saturation temperature.

Fig. 2: Different regimes of boiling. Heat flux data from an electrically heated platinum wire submerged in water. Point (a) represents the critical boiling point. Reproduced from [11]
In Figure 2, it can be seen the different boiling regimes that can be present when a fluid is in contact with a heated surface. In region II, bubbles begin to form on the surface and are dissipated in the liquid after breaking away from the surface. This region indicates the beginning of nucleate boiling. As the temperature excess rises, bubbles form more rapidly and rise to the surface of the liquid to dissipate.

When the excess temperature reaches the point (a), bubbles start forming so rapidly that they blanket the heating surface and prevent the inflow of fresh liquid from taking their place, forming a vapor film that covers the surface. Thermal resistance of this film causes a reduction in heat flux. After this regime, if excess temperature keeps increasing, there is a transition to stable film boiling where a significant portion of the heat lost by the surface may be the result of radiation, and this explains the new increment in the heat flux [11].

The focus on this project is to study the nucleate boiling, since this is the phenomena that can be present in the cooling application for internal combustion engines, and it is a desired design parameter due to the important increment of the heat transfer coefficient and the heat flux, that will help the cooling process to be more efficient.

In the nucleate boiling region (II and III in Figure 2), bubbles are created by the expansion of entrapped gas or vapor at small cavities in the surface. Depending on the temperature excess, the bubbles may collapse on the surface, may expand and detach from the surface to be dissipated in the body of the liquid, or at higher temperatures may rise to the surface of the liquid before being dissipated.

The primary mechanism of heat transfer when nucleate boiling is present is the intense agitation at the heat transfer surface, which creates the high heat transfer rates observed in boiling [11].

In the AVL Fire code there is implemented a module that takes into account the effect of boiling on heat transfer in a water-glycol mixture, this module is called Single-Phase Boiling, it is important to mention that multiphase boiling is not considered in this module.

For the implementation of the Single-Phase Boiling module two different models can be selected, the Chen Boiling model or the Boiling Departure Lift-off (BDL) model. The boiling heat flux associated to nucleate boiling phenomena is modeled as shown in Eq.31:

\[ q_{boiling} = h_{boil}(T_w - T_{sat}) \]  \hspace{1cm} (Eq.31)

where \( h_{boil} \) refers to the microscopic heat transfer coefficient, \( T_{sat} \) to saturation temperature of the liquid, and \( T_w \) to the wall temperature.

As mentioned previously, for the cooling of internal combustion engines, the coolant used is generally a mixture of water and glycol, and depending on the application different glycol concentrations can be used. Figure 3 shows the saturation properties (saturation pressure and saturation temperature) of engine coolant for different glycol concentrations.
2.4.2 Chen Boiling Model

The Chen model is a known boiling model to study the formation of nucleate boiling of flows in contact with heated surfaces. In this model, microscopic boiling heat transfer (without convection) coefficient is defined in Eq.32 [8] [15]:

$$h_{boil} = S_{Chen}h_{pool} \quad \text{(Eq.32)}$$

where the pool boiling heat transfer coefficient is computed with Eq.33:

$$h_{pool} = 0.00122 \left[ \frac{c_p}{\rho_l} \lambda^{0.45} \rho_l^{0.49} \rho_v^{0.24} \sigma^{0.5} \mu^{0.29} \right] \Delta T_{sat}^{0.24} \Delta P_{sat}^{0.75} \quad \text{(Eq.33)}$$

Where $c_p$ is liquid mixture specific heat, $\lambda$ is liquid mixture thermal conductivity, $\rho_l$ is the liquid mixture density, $\rho_v$ is the vapor density, $L_v$ latent heat of vaporization, $\sigma$ is the bubble surface tension, $\mu$ is the liquid mixture dynamic viscosity, $\Delta T_{sat}$ is the local difference between wall temperature and boiling temperature, and $\Delta P_{sat}$ is local difference between the saturation pressure and the static pressure.

Chen’s concept was originally proposed for saturated boiling flow of water. Campbell and his team (1995) were the first group who considered subcooled boiling flow in automotive cooling systems and carried out experiments with engines coolant (ethylene-glycol and water mixtures) and they adopted Chen’s ansatz for the heat flux modelling [5].

Based on their experimental results they proposed the following correlations (Eq.34) for the flow induced suppression factor [5]:

$$S_{Chen} = \begin{cases} 1 & \quad Re < 1.10^4 \\ 3.4 - 0.6 \log(Re) & \quad 1.10^4 \leq Re \leq 4.10^5 \\ 0.04 & \quad 4.10^5 < Re \end{cases} \quad \text{(Eq.34)}$$

The Chen-type model of Campbell (1995) provides a good description of the wall heat transfer enhancement due to nucleate boiling. On the negative side, this model, like most of the proposed superposition models, is not well suited for application in the CFD of coolant.
flows in engineering devices, because the model correlation for the suppression factor depends on the bulk flow Reynolds number as a non-local parameter.

Moreover, dealing with CFD of geometrically complex flow configurations like those in engine coolant jackets, it is practically not possible to define a Reynolds number based on bulk flow conditions in a meaningful way [5].

2.4.3 Boiling Departure Lift-off Model

The Boiling Departure Lift-off (BDL) model is an improvement to the Chen model, which accounts for the dynamic effect of the near-wall flow field on the bubble detachment from the heated surface. Two suppression factors are implemented in this model as shown in Eq.35:

\[ h_{boil} = S_{BDL1} S_{BDL2} h_{pool} \]  
(Eq.35)

First suppression Factor (\(S_{BDL1}\))

The first suppression factor is defined in Eq. 36:

\[ S_{BDL1} = \left( \frac{d_B}{d_L} \right)^n \]
(Eq.36)

where \(d_B\) and \(d_L\) are the departure diameter and the lift-off diameter of the bubble respectively, and they can be determined by force balances at the instants of bubble departure from the nucleation site and bubble lift off from the surface, respectively (see Figure 4 and Figure 5). This procedure is described in “A wall heat transfer model for subcooled boiling flow” [4] and it has been introduced into the AVL Fire code where it performed well in simulations of automotive coolant jackets [5].

In this suppression factor implemented in the AVL Fire code, the exponent “n” in Eq.36 is calculated by AVL from model calibration.

![Fig. 4: Bubble Departure (Nucleation Site) and Lift-off. Reproduced from [8]](image)
Fig. 5: Forces applied on a boiling bubble. Growth force ($F_{du}$), Buoyancy force ($F_b$), Quasy-steady drag force ($F_{qs}$), Shear lift force ($F_{sl}$). Reproduced from [8]

**Second suppression Factor ($S_{BDL2}$)**

Steiner extended the BDL model to the boiling of pure water, where they introduced an additional suppression factor (Eq.37) for the nucleate pool boiling component to capture explicitly the influence of the sub-cooling [4]:

$$S_{BDL2} = \frac{T_w - T_s}{T_w - T_b}$$  \hspace{1cm} (Eq.37)

where $T_w$ is the wall temperature, $T_s$ is the saturation temperature of the liquid and $T_b$ is the flow bulk temperature.

**2.4.4 Transition to film boiling**

Chen-type models do not provide a correct description of the heat flux saturation that occurs when $\Delta T_{sat}$ increases and the fluid enters in the transition boiling region (region IV in Figure 2). In that case, the Chen correlation predicts a continuously increasing heat flux instead of the physical saturation [8].

To deal with this, a critical heat flux boiling coefficient ($h_{crit}$) is specified as an upper limit for the BDL boiling module. This critical heat transfer coefficient is obtained from experimental results in similar coolants used in the automotive industry (water-glycol mixture) on coolant jackets and its value is around the 20000 [W/m$^2$K] [8].

**2.5 Polyhedral Mesh**

A polyhedral mesh corresponds to a 3D partition of the domain into polyhedral cells generally of 12 and 14 sides, although the number of faces is unrestricted. This type of partition claims to be the best way of partitioning space, proposed by physicist Denis Weaire and Robert Phelan, and it is 0.3% more efficient than a complete 14 sided, tetrakaidekahedron cells arrangement, proposed by Lord Kelvin in 1887 [16].

Polyhedral offer the same level of automatic mesh generation as tetrahedral do, while overcoming disadvantages adherent to tetrahedral meshes. Some of the disadvantages of the tetrahedral meshes are that tetrahedral elements have only four neighbors, not being an optimal choice for CFD as computation of gradients at cell centers can become problematic, increasing the chances of stability issues, reduced accuracy and problematic convergence properties. [2] [16]
Tetrahedron, as the lowest order polyhedron, fill space less efficiently than any other element, while a mesh consisting of polyhedral cells has fewer faces than any other cell type for a similar geometry, having an important impact on the solver efficiency \[16\] \[2\].

An important advantage of using polyhedral cells comes from the fact that they are bounded by more neighbors than traditional cell types (Figure 6). As mentioned, tetrahedral cell communicates with only four neighbor cells, and a hexahedral with six, limiting the approximation of gradients for each cell. On the other hand, polyhedral cells have on average 12 or 14 neighbors, allowing information to propagate faster through a polyhedral mesh, leading to an increased rate of convergence \[2\] \[16\].

![Triangular element adjacencies](a) Triangular element adjacencies  
![Polygonal element adjacencies](b) Polygonal element adjacencies

**Fig. 6:** Neighboring elements for triangular and polygonal meshes. Reproduced from [2]

Comparisons and many practical tests have concluded that polyhedral meshes need about four times fewer cells, half the memory and a tenth to fifth of computing time compared to tetrahedral meshes to reach solutions of the same accuracy, having better convergence properties \[1\].

As an example, CD-adapco performed tests on the water cooling jacket of an engine, similar to the present project, comparing polyhedral and tetrahedral grids. Results obtained on any polyhedral mesh are more accurate than results obtained on tetrahedral mesh with a comparable number of cells. In their test, the results from a polyhedral mesh with \(6 \times 10^4\) cells is slightly more accurate than the result from a tetrahedral mesh with \(39 \times 10^4\) cells (6 times more). Computing time on this polyhedral mesh is less than one tenth of the computing time for the tetrahedral mesh that would deliver same accuracy \[1\].

Negative comments have arisen for the polyhedral grids, claiming that this meshes require a considerable amount of adjacent relations in comparison to tetrahedral and hexahedral meshes, having the need of resource expensive solutions \[2\]. However, research has been made in polyhedral grids generation methods towards more efficient and high quality meshes, and CFD software have started to incorporate this type of mesh generation tool in their code \[1\] \[16\].

Such is the case of AVL Fire with the incorporation a new grid generation solution called FAME Poly, a fully automated meshing environment for complex geometries, being able to generate high quality polyhedral meshes. This new solution is one of the reasons and motivation for this project to be developed at Scania CV AB, user of the AVL Fire software for engine CFD analysis, where the advantages of using this new type of grids will be tested and implemented in a new simulation method for thermal analysis of engine components.
3. Method

The methodology to fulfill the thesis objectives involves two main stages. The first stage consists of the study of a simplified Scania engine geometry within the software AVL Fire M and AVL Fire classic, using simplified boundary conditions in order to test the new procedure.

The second stage corresponds to a study of a complete geometry of 1-cylinder, from a SCANIA 6-cylinder engine, with the corresponding non-simplified boundary conditions and a comparison with former methodology results and experiments, in this stage a mesh sensitivity analysis is performed.

The engine to be studied or analyzed during this project corresponds to the SCANIA DC13, 6 cylinders in-line with 450 hp, diesel type. It is a 13 liters’ engine that fulfills Euro 6 truck engine certifications [17] and it is shown in Figure 7.

![DC13, 6 cylinder, 450 hp engine. Shown with permission from Scania CV AB](image)

**Fig. 7:** DC13, 6 cylinder, 450 hp engine. Shown with permission from Scania CV AB [18]

The CAD geometry to be used during this project can be seen in Figures 8 and 9, this geometry corresponds to one cylinder of the mentioned engine, and the parts to be modelled are the cylinder head, the cylinder block, the water cooling jacket, the liner, the intake and exhaust valves and seats, and the guides of the valves. Parts have been presented and described in Section 2.1.1.
Fig. 8: DC13 SCANIA CAD geometry. Full 1-cylinder model assembly (blue). Details of different domains shown: Intake valves (dark blue), Exhaust valves (red), Guides (metallic), Intake seats (light brown), Exhaust seats (dark brown), Water cooling jacket (aquamarine), Cylinder Liner (dark grey), Gasket (light grey).

Fig. 9: DC13 SCANIA CAD geometry assembly details. Parts have been cut along the engine center line (Not all parts shown in the figure). Shown with permission from Scania CV AB [14].

This study has the characteristic of dealing with turbulent flow, explained in the theory in Section 2.2.2, like in many industrial applications, with a Reynolds number at the inlet of the water cooling jacket section of the cylinder head being around 77000, so a Reynolds Average Navier-Stokes (RANS) turbulent model is implemented. Computation of the Reynolds
number, with diameter as reference length, for the inlets and outlet of the water cooling jacket of the cylinder head (Figure 10), are shown in Table 2.

![Figure 10: Water cooling jacket section of the Cylinder head. Inlet 1, Inlet 2, and Outlet for the coolant are shown. Reynolds number is calculated in these sections.](image)

**Tab. 2:** Reynolds values for the water cooling jacket domain (Diameter as reference length)

<table>
<thead>
<tr>
<th></th>
<th>Red [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet 1</td>
<td>77291</td>
</tr>
<tr>
<td>Inlet 2</td>
<td>21067</td>
</tr>
<tr>
<td>Outlet</td>
<td>74741</td>
</tr>
</tbody>
</table>

The turbulence model to be used corresponds to the $\zeta$-f model explained in the theory, Section 2.2.4, this is the recommended turbulence model for these applications (engine conjugate heat transfer problems) according to the developers of the AVL Fire code, and superior to the other RANS models implemented in the software like the $k$-$\varepsilon$ model.

The simulations to be performed during this project are steady-state simulations, since the boundary conditions used are not time dependent values and it is assumed that the engine is being operated at a constant regime or operating condition. By these simulations, it is possible to obtain the distribution of thermal properties of engine components at the specific operating point at which the combustion simulation is performed. It is of relevance to mention that the combustion simulation results are time-averaged within one engine cycle, to be able to use them as boundary conditions for thermal steady state simulations.

The different materials corresponding to each domain or part in our model, are characterized by having a temperature dependent thermal conductivity property, an important fact to consider when working in a wide range of temperatures to capture the correct thermal behavior in the simulations.

In this project, the convection heat transfer is characterized by being forced convection with the presence of boiling for the domain of the water cooling jacket inside the internal combustion engine. According to the theory, Section 2.3.2, the reason for having forced convection comes from the fact that the presence of velocity in the fluid, in this case the coolant, is due to external sources, and not a result of density gradients related to temperature variations of the fluid next to the wall.

As explained in Section 2.4, the boiling module is important to be included in this type of studies, since formation of nucleate boiling in the water cooling jacket of the engine implies a
high increment in the heat transfer. The boiling module to be used for these simulations corresponds to the BDL model.

To use the BDL model in AVL Fire, the critical boiling heat flux coefficient ($h_{\text{crit}}$) needs to be specified as an upper limit for the boiling module. This critical heat transfer coefficient is obtained from experimental results in similar coolants used in the automotive industry (water-glycol mixtures) on coolant jackets and the value to be used is set to be 20000 [W/m²°C] [8].

Radiation effects can be neglected in this study, as mentioned in Section 2.3.3 the thermal phenomena are dominated by forced convection and conduction of heat in the model being analyzed and compared [12] [13]. Therefore, no radiation is included in the simulations.

3.1 Stage 1: Simplified SCANIA engine geometry study

The geometry model of the engine cylinder to study during this stage includes the cylinder head, the intake and exhaust valves, the intake and exhaust seats, the guides and the cylinder head section of the water cooling jacket. The cylinder block, water cooling jacket part of the block, and the liner are not included in this stage to simplify the process.

Another important simplification in this stage is the use of space average boundary conditions obtained from previous combustion simulations, instead of the direct mapping of this results into our simulations. The main objective of this first stage is to test the procedure of the implementation of the new methodology with the new features related to the use of a polyhedral mesh and the automatic identification of domain interfaces. In this first stage, it is not intended to compare the physic of fluid dynamic results directly with the previous methodology, due to the simplification implemented in the geometry and the fact that different boundary conditions (average values) are used.

3.1.1 Pre-processing stage

During the pre-processing stage the CAD model must be prepared, this is done in a preprocessing Computer-aided engineering (CAE) tool named ANSA from BETA CAE Systems, where basically double faces between material interfaces are removed and edges are merged as required by the meshing tool to be used in AVL Fire M, this corresponds to a careful and time consuming process.

It is important to mention that all surfaces face selections are defined in this step within the ANSA software, this is a procedure that will allow the user to define groups of domains (e.g. group of all the guide components in one unique domain), and assign mesh discretization settings during the mesh generation, as well as to assign the boundary conditions to the corresponding faces during the setup stage.

After all the face selections are set, it is possible to proceed to generate a .stl representation of the geometry, this file will serve a base to the creation of the polyhedral mesh within the AVL Fire M software.

In order to check the geometry prepared in ANSA, it is recommended to run a Geometry Check analysis before continuing to further stages of the study. This geometry check allows to identify errors in the geometry that must be fixed, like the presence of faces intersections, gaps in the geometry, overlap faces and needle faces. This is an iterative process of preparation of the geometry, were the engineer fixes geometrical errors and generate the .stl representation until the geometry check analysis is passed. Visualization of the geometry with assigned face selections together with the geometry checks performed in ANSA are shown in the Appendix. Once the CAD model is ready for volume meshing, it is imported into the meshing tool of AVL Fire M.

In the AVL Fire M software, first it is needed to run a surface repair analysis, by doing this, the imported .stl Geometry representation from ANSA will be analyzed to fulfill specific requirements of the software, like the presence of duplicated elements, cracks, self-intersections and overlaps, holes and orientation of the faces. If the quality of the imported .stl
geometry is good, reparations can be done with the automatic tools included in AVL Fire M, and manual reparation can be avoided.

It is relevant to mention that the AVL Fire M meshing tool (FAME POLY) identifies domains within closed surfaces, and faces that are shared between two different domains are automatically defined as interfaces.

After the surface repair process, an edge definition must be created, this means to define all the important edges that will serve as a guide for the meshing process. It is important to define the edges specifically in the interfaces of the domains, to clearly define the boundaries of each part or component. For this step, it is possible to use the automatic edge detection tool in AVL Fire M, followed by a manual check to add or delete edges considered by the engineer. This will guarantee that the volume mesh reproduces accurately the geometry to be analyzed.

For the simplified geometry study, meshing discretization parameters are shown in Table 3. Values assigned to target surface cell size and target volume cell size are similar to discretization mesh settings implemented in the former ACCI methodology used at Scania CV AB.

The inflation layers’ settings for the water cooling jacket domain in this first stage are set to have only one inflation layer with a 0.5 mm of total thickness, this setting represents a first guess, following the settings from the former method and example simulations from the AVL Fire tutorials in engine water cooling jackets. In this stage, the aim is to obtain relatively low $y^+$ values, between 1 and 30, without the need of generating an extremely refined mesh that will demand more computational resources. This inflation layer setting is applied knowing that the hybrid wall treatment explained in Section 2.2.5 will be used. Details on this wall treatment are commented in the set-up stage of this method.

**Tab. 3: Mesh discretization settings for different domains of the simplified Scania geometry.**

<table>
<thead>
<tr>
<th>Part/Domain</th>
<th>Target Surface cell size [mm]</th>
<th>Target volume cell size [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exhaust Seats</td>
<td>1</td>
<td>1.5</td>
</tr>
<tr>
<td>Intake Seats</td>
<td>1</td>
<td>1.5</td>
</tr>
<tr>
<td>Guides</td>
<td>1</td>
<td>1.5</td>
</tr>
<tr>
<td>Exhaust Valves</td>
<td>1</td>
<td>1.5</td>
</tr>
<tr>
<td>Intake Valves</td>
<td>1</td>
<td>1.5</td>
</tr>
<tr>
<td>Cylinder Head</td>
<td>3</td>
<td>5</td>
</tr>
<tr>
<td>Water Cooling Jacket</td>
<td>1.5</td>
<td>2</td>
</tr>
</tbody>
</table>

One objective of this initial stage is to test the methodology and the implementation of the polyhedral mesh. The mesh dependency study is to be performed in the next stage with the complete geometry of the engine cylinder.

To evaluate the quality of the mesh, the parameters to study are the skewness of the cells and the orthogonality, in this project the goal is to generate meshes without any cells having skewness values higher than 0.95 (where 1 is the worst) or orthogonality values less than 0.1 (where 0 is the worst). These parameters are commonly used in the CFD community to obtain meshes with a lower risk of divergence. The generated polyhedral mesh for the simplified geometry stage is shown in Figure 11.
Fig. 11: Polyhedral mesh for the simplified SCANIA engine geometry.

No skewness, orthogonality problems, nor negative volume cells are found in the generated mesh and domain interfaces have been automatically created.

3.1.2 Setup and Simulation

During the setup, the materials of each domain are specified, including temperate dependent thermal conductivity values explained in the theory (Section 2.3.1), together with solver settings and solver control options. Materials assigned for each part or component in this model are shown in Table 4 and are obtained from the engine specifications available at the working department.

Tab. 4: Materials applied for each engine component during the setup for the simple Scania geometry. These materials have varying properties with temperate that are specified in the AVL Fire software.

<table>
<thead>
<tr>
<th>Part</th>
<th>Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exhaust Seats</td>
<td>AR20 (Steel alloy)</td>
</tr>
<tr>
<td>Intake Seats</td>
<td>PL12 (Steel alloy)</td>
</tr>
<tr>
<td>Guides</td>
<td>GJL300 (Cast Iron)</td>
</tr>
<tr>
<td>Exhaust Valves</td>
<td>X50CrMnNiNbN21 (Stainless steel – heat treatment)</td>
</tr>
<tr>
<td>Intake Valves</td>
<td>Nimonic (nickel-based high-temperature low creep super alloys)</td>
</tr>
<tr>
<td>Cylinder Head</td>
<td>GJL300 (Cast Iron)</td>
</tr>
<tr>
<td>Water Cooling Jacket</td>
<td>Water 60%/Glycol 40% (Engine coolant)</td>
</tr>
</tbody>
</table>
In this simplified geometry stage, the boundary conditions correspond to space average values of heat transfer parameters taken from existing combustion simulations for the same engine, these are space averaged heat transfer coefficients (HTC) and space averaged temperatures, as it can be seen in Table 5. In the next stage, the temperature values and heat transfer coefficients will be mapped directly as the results from combustion simulations for each corresponding face, to obtain the most realistic results and be able to compare with previous thermal simulations (former ACCI method). The combustion boundary conditions used correspond to values from simulations performed at an engine operating point of 1800 rpm with 115% of load.

**Tab. 5:** Boundary conditions for simplified Scania geometry. Average values for heat transfer coefficient (HTC) and temperatures. Obtained from existing combustion simulations at Scania CV AB (1800 rpm / 115% load).

<table>
<thead>
<tr>
<th>Part</th>
<th>Surface</th>
<th>Mean HTC [W/m²K]</th>
<th>Mean Temp. [K]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seats</td>
<td>Exhaust Seats - Chamber</td>
<td>1880</td>
<td>624</td>
</tr>
<tr>
<td></td>
<td>Intake Seats - Chamber</td>
<td>1179</td>
<td>440</td>
</tr>
<tr>
<td>Guides</td>
<td>Exhaust Guides - Chamber</td>
<td>707</td>
<td>808</td>
</tr>
<tr>
<td></td>
<td>Intake Guides - Chamber</td>
<td>312</td>
<td>353</td>
</tr>
<tr>
<td>Valves</td>
<td>Exhaust Valves - Chamber</td>
<td>2239</td>
<td>992</td>
</tr>
<tr>
<td></td>
<td>Intake Valves - Chamber</td>
<td>1577</td>
<td>602</td>
</tr>
<tr>
<td>Cylinder</td>
<td>Exhaust Port</td>
<td>683</td>
<td>721</td>
</tr>
<tr>
<td>Head</td>
<td>Fire deck</td>
<td>1861</td>
<td>677</td>
</tr>
<tr>
<td></td>
<td>Intake port straight</td>
<td>346</td>
<td>345</td>
</tr>
<tr>
<td></td>
<td>Intake port curved</td>
<td>345</td>
<td>345</td>
</tr>
<tr>
<td></td>
<td>Exterior surface</td>
<td>50</td>
<td>298</td>
</tr>
<tr>
<td></td>
<td>Valve train</td>
<td>50</td>
<td>379</td>
</tr>
<tr>
<td></td>
<td>Oil passage</td>
<td>1000</td>
<td>378</td>
</tr>
<tr>
<td></td>
<td>Fuel rail</td>
<td>1000</td>
<td>343</td>
</tr>
<tr>
<td></td>
<td>Saver ring top</td>
<td>472</td>
<td>602</td>
</tr>
</tbody>
</table>

Boundary conditions and properties definitions are also needed for the coolant, as the fluid to be used inside the water cooling jacket domain as part of the cooling system of internal combustion engines described in the theory. The coolant corresponds to a water glycol mixture that behaves as an incompressible fluid in the range of operation where fluid dynamic equations are simplified for this incompressible condition as explained in Section 2.3.4. Parameters and properties of the coolant are shown in Table 6.
Tab. 6: Boundary conditions and properties for the coolant of the water cooling jacket domain. Conditions correspond to engine operating point of 1800 rpm with 115% of load.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coolant glycol fraction</td>
<td>0.4</td>
</tr>
<tr>
<td>Inlet Boundary condition</td>
<td>Mass flow: 0.91 kg/s, Temp.: 90°C</td>
</tr>
<tr>
<td>Outlet Boundary condition</td>
<td>Static pressure: 2.82 bar</td>
</tr>
<tr>
<td>Deairation ports Boundary condition</td>
<td>Mass flow: -0.01 kg/s</td>
</tr>
<tr>
<td>Density</td>
<td>1016 kg/m³</td>
</tr>
<tr>
<td>Reference temperature</td>
<td>298.15 K</td>
</tr>
</tbody>
</table>

Simulations to perform correspond to RANS simulations, and the $\zeta$-$f$ turbulence model is the one to be used, explained in Section 2.2.4. Differencing scheme used for momentum and continuity equations is central difference, and for turbulence and energy equation the first order upwind scheme is used, these are recommended and default schemes of the AVL Fire solver [8], also used in the former methodology for thermal simulations with satisfactory results. Converge criteria used for the different equations is the common approach used in thermal simulations at Scania, to guarantee a converged solution [14]. Solver settings are shown in Table 7.

Tab. 7: AVL Fire Solver settings for Stage 1 simulations.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simulation type</td>
<td>Steady-state RANS</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>$\zeta$-$f$</td>
</tr>
<tr>
<td>Wall treatment</td>
<td>Hybrid Wall treatment</td>
</tr>
<tr>
<td>Under relaxation factors</td>
<td>Momentum: 0.6</td>
</tr>
<tr>
<td></td>
<td>Pressure: 0.1</td>
</tr>
<tr>
<td></td>
<td>Turb. kin. energy: 0.4</td>
</tr>
<tr>
<td></td>
<td>Turb. diss. rate: 0.4</td>
</tr>
<tr>
<td></td>
<td>Energy: 0.6</td>
</tr>
<tr>
<td>Differencing scheme</td>
<td>Momentum: Central difference</td>
</tr>
<tr>
<td></td>
<td>Continuity: Central difference</td>
</tr>
<tr>
<td></td>
<td>Turbulence: Upwind</td>
</tr>
<tr>
<td></td>
<td>Energy: Upwind</td>
</tr>
<tr>
<td>Convergence criteria</td>
<td>Max. Iterations: 20000</td>
</tr>
<tr>
<td></td>
<td>Momentum: 1e-5</td>
</tr>
<tr>
<td></td>
<td>Pressure: 1e-5</td>
</tr>
<tr>
<td></td>
<td>Turb. kin. energy: 1e-5</td>
</tr>
<tr>
<td></td>
<td>Turb. diss. rate: 1e-4</td>
</tr>
<tr>
<td></td>
<td>Energy: 1e-8</td>
</tr>
</tbody>
</table>
It is important to mention that to achieve a stable solution, it was needed to run several simulations while adjusting the under-relaxation factors for the governing equations and applying the recommendations from AVL Fire support regarding the single-phase boiling module, which may have divergence problems in the energy equation. Stability problems when using the single-phase boiling module have been an issue within the Scania NMGD department when applying the former ACCI methodology.

Recommendations from AVL Fire support regarding the stability of the boiling module are to activate it after several iterations (e.g. 500) or from a partially converged solution, and to evaluate the static enthalpy instead of the total enthalpy for the energy equation in the water cooling jacket domain [8]. These recommendations increased significantly the stability of the solution, making the simulations able to converge.

An important numerical model to include in the methodology is the Hybrid wall treatment explained in Section 2.2.5, available in the AVL Fire code. This wall treatment allows having values of $y^+$ in the three different regions of the boundary layer being these ones the viscous-sub layer for $y^*<5$, the buffer layer for $5<y^*<30$ and the fully turbulent region for $y^*>30$; where the solver will apply either the integration of the turbulence model up the wall, the use blending functions or the use of wall functions, respectively. As mentioned during the mesh generation step, the aim is to have $y^+$ values between 1 and 30, the reason behind this is that for higher values of $y^+$, with the use of wall functions, the solution starts to differ from reality due not accurately model the flow inside the boundary layer [9]. In the mesh dependency study to perform in the next stage, different settings for the inflation layer will be tested to evaluate the dependency of the $y^+$ values and the behavior of the hybrid wall treatment. Figure 12 shows the $y^+$ values obtained for the water cooling jacket in the polyhedral mesh generated in the first stage.

**Fig. 12:** $y^+$ values on the Water Cooling Jacket for Scania simple geometry. From $y^+$ values it is possible to understand how the hybrid wall treatment is used in the simulations.

A relevant feature of the new methodology, is the possibility to implement thermal contact resistance values between the different parts or domains as mention in the theory Section 2.3.1, this allows to account for the effect of displacement of the valves through the guides and the cyclic contact between valves and seats (opening and closing of valves), which is not possible to directly simulate it in a steady state analysis, since this analysis is not time dependent and no moving mesh is used.

In the former method, the default value between parts or domains interfaces for the contact thermal resistance is set to be zero, and there is no possibility to change it. By implementing the new methodology, the thermal contact resistance between materials can be specified and, in that sense, reproduce more accurate results. Implementation of thermal contact resistance
between the parts plays an important role in the heat transfer between them, for example, valves and seats, that may affect the overall results of the simulation.

In the new method, the thermal contact resistance between valves and seats is set to be $1 \times 10^{-4}$ [mK/W], and between valves and guides to be $1 \times 10^{-5}$ [m²K/W]. These values are set following the recommendation from AVL Fire support, to consider the cyclic contact between the parts (opening and closing of the valves) while using steady state simulations [8].

After the simulations are performed, a post-processing work will provide important data to validate the model and the method, like temperature distribution along the engine parts, conditions of the coolant, presence of boiling, etc. Results will be compared with existing data from the former ACCI methodology. For this step, it is important to remark that simplified boundary conditions are implemented, so the goal in the comparison of the results is to test the overall method and not to validate specific fluid dynamics results.

### 3.2 Stage 2: Complete SCANIA Geometry (1-cylinder)

The methodology to follow during this phase is similar to the first stage, implementing the multi-material options for the polyhedral mesh generation, adding the single-phase boiling model, and using the $\zeta$-t turbulence model within the simulations. In this phase complexity is added to the geometric model by including the cylinder block, the liner, and the water cooling jacket region corresponding to the block. The complete cylinder is being modeled and simulated, implementing boundary conditions mapped directly from the combustion simulations, to be able to compare results directly with the former methodology and with available experimental results.

#### 3.2.1 Pre-processing stage

During the pre-processing stage, the full 1-cylinder geometry is prepared in the preprocessing tool ANSA, adding the corresponding face selections to all the added surfaces (cylinder block, liner, gasket and saver ring). As explained in the first stage, geometry checks are done and passed prior to exporting the .stl geometry representation file to AVL Fire M for the polyhedral mesh generation. The surface repair process in AVL Fire M is carried out with the automatic repairation tools available, and all surface repair checks are passed to continue with the generation of a polyhedral mesh. Figures showing the prepared geometry with assigned face selections, together with the geometry checks performed in ANSA can be found in the Appendix.

A mesh dependency study is carried out in this stage, where different mesh discretization settings are proposed to obtain information regarding the best cell size and inflation layer settings to be used in this type of thermal simulations when working with polyhedral meshes. During the mesh dependency study, it is decided that the domain to be altered in elements size is the fluid domain (water cooling jacket domain), since this is the most sensible domain of the mesh due to the amount of physics involved, accounting for all the fluid dynamics variables like velocity and pressure, including thermal variables like temperature and heat transfer coefficients as mentioned in the Theory (Section 2.2). In the fluid domain, the CFD equations presented in Section 2.2 must be solved, these are conservation of mass, of momentum and energy; meanwhile, in the solid domains, only the energy equation is needed to be solved to obtain the distribution of temperature along the parts as explained in Section 2.3.4, so cells size in the solid domains will play a less important role in the convergence of the solution.

The mesh dependency study is performed in two steps, the first one is to test the impact of the global cell size in the solution, using constant values for the settings of the inflation layer. The variation of the overall cell size in the water cooling jacket is done by implementing changes of ±30% for both surface and volume target cell size from a base mesh.

The base mesh is the starting grid for the mesh sensitivity study, and corresponds to a grid that has the same mesh settings used for the first stage, having also similar settings to the mesh...
used in the former methodology to compare with (ACCI methodology), representing a good starting point for the study. Mesh settings for the global cell size study are shown in Table 8.

**Tab. 8:** Mesh dependency study. WCJ global cell size settings. Number of cells have been normalized with the base mesh used as reference.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>WCJ target surface cell size [mm]</th>
<th>WCJ target volume cell size [mm]</th>
<th>WCJ inflation layers [-]</th>
<th>Inflation layer thickness [mm]</th>
<th>WCJ number of cells [-]</th>
<th>Total number of cells [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base Mesh</td>
<td>1.5</td>
<td>2</td>
<td>1</td>
<td>0.5</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>Coarse Mesh</td>
<td>1.95</td>
<td>2.6</td>
<td>1</td>
<td>0.5</td>
<td>0.60</td>
<td>0.89</td>
</tr>
<tr>
<td>Fine Mesh</td>
<td>1.05</td>
<td>1.4</td>
<td>1</td>
<td>0.5</td>
<td>2.32</td>
<td>1.35</td>
</tr>
</tbody>
</table>

The second step corresponds to test the impact of changing the inflation layers’ parameters to vary the $y^*$ values, and study the impact of either integrate the turbulence model to the wall ($y^*$<5), the use of blending functions ($5<y^*$<30), and the use of wall functions ($y^*$>30), according to the hybrid wall treatment being used, and explained in Section 2.2.5. For the inflation layer settings study, the considered independent mesh from the first step is used as the starting mesh, where global cell size settings will be fixed.

It is relevant to mention that the mesh independency study is performed with the settings explained in the Setup and simulation section (Section 3.2.2), mapping boundary conditions directly from combustion simulations for a specific engine operating point of 1800 rpm and 115% of the load, these conditions represents an extreme operating point for the engine and it is a common setting for experimental tests at Scania CV AB [14]. The coolant characteristics correspond to a water/glycol mixture with a 0.4 glycol fraction.

**Tab. 9:** Mesh dependency study. Global cell size variation results. Pressure diff. % is computed for the immediate previous mesh in each case. All values have been normalized with the base mesh used as reference.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>WCJ cells</th>
<th>Total Heat</th>
<th>Conv. Heat</th>
<th>Boil Heat</th>
<th>Boil Area</th>
<th>Temp diff.</th>
<th>Pressure drop</th>
<th>Pressure diff. [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse Mesh</td>
<td>0.60</td>
<td>1.00</td>
<td>0.997</td>
<td>1.019</td>
<td>1.041</td>
<td>0.999</td>
<td>1.078</td>
<td>-</td>
</tr>
<tr>
<td>Base Mesh</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>1.00</td>
<td>-7.49</td>
</tr>
<tr>
<td>Fine Mesh</td>
<td>2.32</td>
<td>0.999</td>
<td>1.016</td>
<td>0.916</td>
<td>0.953</td>
<td>0.996</td>
<td>0.953</td>
<td>-4.28</td>
</tr>
</tbody>
</table>

Analysis of the results from the first step in the mesh sensitivity study shows no mayor differences between the variety of mesh sizes as it can be seen in Table 9, where similar values are obtained for the total heat transferred, for the proportions of boiling and convective heats, and the for boiling area inside the water cooling jacket. The difference that can be seen between the Boiling Heat for the Base and the Fine meshes, of less than 10%, is considered to be very small for this parameter, especially when moving forward to study the impact on this field in the inflation layer study, where differences are in the order of 50%. Due to not having great variations in the total heat, other results of interest like the temperature difference of the coolant also shows no significant changes between different meshes.

Pressure drop value is of interest and helpful to study mesh dependency in this case, since it is a parameter that is not linked to the boiling module like the rest of the parameters, pressure drop percentage differences are calculated based on the immediate previous mesh for each case, to study the change on this parameter between consecutive meshes. Values for the pressure drop inside the WCJ for different cell sizes are also shown in Figure 13, where it can be seen a converge behavior.
Fig. 13: Pressure drop inside the engine water cooling jacket for one cylinder. Results from the global cell size study of the mesh sensitivity analysis. Pressure drop values show a converge behavior with less than a 5% difference between the base mesh and the fine mesh.

From the global cell size study, the base mesh is chosen as the starting mesh for the inflation layer setting analysis, since it can be concluded that it has a low global size mesh sensitivity. The different inflation layer parameters for the second step in the mesh sensitivity study are shown in Table 10.

Tab. 10: Mesh dependency study. WCJ inflation layer settings. Number of cells have been normalized with the base mesh used as reference.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>WCJ target surface cell size [mm]</th>
<th>WCJ target volume cell size [mm]</th>
<th>WCJ inflation layers [-]</th>
<th>Inflation layer thickness [mm]</th>
<th>WCJ number of cells [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>High $y^+$</td>
<td>1.5</td>
<td>2</td>
<td>1</td>
<td>1</td>
<td>0.95</td>
</tr>
<tr>
<td>Base Mesh</td>
<td>1.5</td>
<td>2</td>
<td>1</td>
<td>0.5</td>
<td>1.00</td>
</tr>
<tr>
<td>Low $y^+$</td>
<td>1.5</td>
<td>2</td>
<td>3</td>
<td>0.5</td>
<td>1.59</td>
</tr>
<tr>
<td>Lower $y^+$</td>
<td>1.5</td>
<td>2</td>
<td>5</td>
<td>0.5</td>
<td>2.37</td>
</tr>
<tr>
<td>Min. $y^+$</td>
<td>1.5</td>
<td>2</td>
<td>7</td>
<td>0.5</td>
<td>3.02</td>
</tr>
</tbody>
</table>

Results from the second step in the mesh sensitivity study can be seen in Table 11, contour plots showing $y^+$ values for different cases correspond to Figure 14.
Fig. 14: $y^+$ values on the water cooling jacket domain for different inflation layers’ settings. Only three cases shown: base mesh, 5 inflation layers’ mesh and 7 inflation layers’ mesh. As the number of inflation layers for the mesh is increased the $y^+$ values obtained are closer to 1.

Tab. 11: Mesh dependency study. Inflation layer settings variation results. Pressure diff. % is computed for the immediate previous mesh in each case. All values have been normalized with the base mesh used as reference.

<table>
<thead>
<tr>
<th></th>
<th>WCJ cells</th>
<th>Total Heat</th>
<th>Conv. Heat</th>
<th>Boil Heat</th>
<th>Boil Area</th>
<th>WCJ Temp diff.</th>
<th>Pressure drop</th>
<th>Pressure diff. [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>High $y^+$</td>
<td>0.95</td>
<td>1.010</td>
<td>1.003</td>
<td>1.048</td>
<td>1.054</td>
<td>1.008</td>
<td>0.969</td>
<td>-</td>
</tr>
<tr>
<td>Base Mesh</td>
<td>1.00</td>
<td>1.000</td>
<td>1.000</td>
<td>1.000</td>
<td>1.000</td>
<td>1.000</td>
<td>1.000</td>
<td>3.23</td>
</tr>
<tr>
<td>Low $y^+$</td>
<td>1.59</td>
<td>1.058</td>
<td>1.163</td>
<td>0.534</td>
<td>0.631</td>
<td>1.014</td>
<td>1.102</td>
<td>10.16</td>
</tr>
<tr>
<td>Lower $y^+$</td>
<td>2.37</td>
<td>1.065</td>
<td>1.173</td>
<td>0.521</td>
<td>0.573</td>
<td>1.028</td>
<td>1.141</td>
<td>3.55</td>
</tr>
<tr>
<td>Min. $y^+$</td>
<td>3.02</td>
<td>1.055</td>
<td>1.152</td>
<td>0.572</td>
<td>0.616</td>
<td>1.010</td>
<td>1.133</td>
<td>-0.68</td>
</tr>
</tbody>
</table>

As it can be seen in Table 11, there is an important difference in the boiling presence when the $y^+$ values on the mesh are lower, significantly reducing the boiling area and the boiling heat fraction of the total heat transferred, having significant changes close to 50% when comparing the 5 layers’ mesh to the base mesh results, the boiling presence is also graphically compared in Figure 15 and Figure 17. This can be explained due to the fact that for a finer mesh next to the walls, a better and accurate representation of the temperature distribution is obtained, as it can be seen in Figure 16.
In Table 11, the total heat transfer remains with small changes for the different meshes, with values of around 6% difference, that is not considered to be relevant or significant, providing similar results for the temperature difference between the inlet and the outlet of the coolant. Similar total heat values are obtained since the heat that is not considered to be from boiling in the finer meshes is transferred via convection, as it can be seen when comparing the convective and boiling heat parameters in the Table.

Reasons that explain why there is not important differences in the total heat are shown in the theory for Single-phase boiling section (Section 2.4). The explanation is that in regions of boiling presence, the temperature differences obtained between the majority of hot walls on the water cooling jacket and the saturation temperatures of the coolant are not high (higher temperatures are concentrated in small specific regions of the WCJ as it can be seen in Figure 16), so the increment in the boiling heat transfer coefficient is small, as it can be seen in Figure 2 (Section 2.4), making it similar to the convective heat transfer coefficient that would be applied in case of no boiling presence.

Figure 15 shows a decrease in the boiling area when the mesh is finer next to the wall and the boundary layer is modelled by the integration of the turbulence model explained in the theory section of this report. These differences are due to fewer areas in the water cooling jacket walls with temperatures higher than the saturation point of the coolant, although there are higher temperature concentrations in specific regions. This behaviour is graphically described in Figure 16.

**Boiling Presence [-]**

![Boiling Presence](image1)

Case: Base mesh, 1 layer

Case: Lower $y^+$, 5 layers

Case: Min. $y^+$, 7 layers

**Fig. 15:** Boiling presence in the water cooling jacket. Presence of boiling is shown in regions of the water cooling jacket where the walls have a temperature higher than the saturation temperature of the coolant.
Fig. 16: Temperature values on the water cooling jacket walls for different inflation layers’ settings. In meshes with high $y^+$ values overall temperatures are more evenly distributed, meanwhile in meshes with low $y^+$ values the temperatures are more concentrated in specific regions of the water cooling jacket. Temperature values are hidden for confidentiality reasons at Scania CV AB. Same temperature scale is used for all cases shown.

As shown in Figure 16, for meshes with high $y^+$ values the overall temperatures distribution presents lower maximum values but with more area above coolant saturation temperature, meanwhile in meshes with low $y^+$ values the temperatures are more concentrated in specific regions of the water cooling jacket, with higher maximum values but a less amount of area with temperatures above coolant saturation value.

In order to complete the mesh sensitivity study, convergence behaviour of the boiling area and the pressure drop inside the water cooling jacket are studied. These parameters are graphically represented in Figures 17 and 18 respectively.

Fig. 17: Boiling area as a percentage of the complete water cooling jacket area. Convergence behavior shown with a tendency of having less boiling area as the mesh gets finer close to the walls (low $y^+$ values).
Fig. 18: Pressure drop inside the engine water cooling jacket for one cylinder. Results from the inflation layer study of the mesh sensitivity analysis. Pressure drop values show a converge behavior with less than 1% difference between the mesh with 5 inflation layers and the mesh with 7 inflation layers.

From this study it is possible to obtain a configuration that has a low mesh sensitivity for different studied parameters as seen in Figure 17 and 18 for independent parameters like boiling region and the pressure drop, providing reliable results. The mesh selected to be used for the following steps of the project is one that has low $y^+$ values according to the converging behavior of the mentioned figures, being selected the configuration with 5 inflations layers and a total thickness of 0.5 mm. This is a mesh that provides accurate temperature distribution in the water cooling jacket walls, as it can be seen in Figure 17, identifying the regions of higher temperatures close to the exhaust ports of the cylinders, meanwhile the temperature distribution in higher $y^+$ meshes is more disperse. For this reason, the boiling features on low $y^+$ meshes provide more accurate results. The “5 inflation layers’ mesh” shows a mesh independency behavior when compared to the “7 inflation layers’ mesh”.

In this study there is an interest to obtain reliable and accurate results not only for the overall heat transfer and the temperature difference on the water cooling jacket, but also accurate temperature distribution along the engine parts, accurate representation of the boiling presence inside the WCJ and reliable results for the pressure drop. These requirements make the mesh with the low $y^+$ values the best option.

Although the selected mesh (5 inflation layers and total thickness of 0.5 mm) has a high amount on cells in the water cooling jacket domain (solid domains remains with initial base settings), it is not a prohibitive number and simulations can still be performed in a reasonable amount of time. The increment in the computational cost pays off since accurate values are obtained for the studied parameters.

Table 12 shows the settings of the selected mesh from the sensibility analysis and Figures 19 and 20 display the generated polyhedral mesh. Refinements are used in the name selections for mapped boundary conditions, with a cell size of 1.5 [mm] and a refinement depth of 3 [mm].
Table 12: Mesh discretization settings for different domains of the complete Scania geometry.

<table>
<thead>
<tr>
<th>Part/Domain</th>
<th>Target Surface cell size [mm]</th>
<th>Target volume cell size [mm]</th>
<th>Inflation layers [-]</th>
<th>Inflation layer thickness [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exhaust Seats</td>
<td>1</td>
<td>1.5</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>Intake Seats</td>
<td>1</td>
<td>1.5</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>Guides</td>
<td>1</td>
<td>1.5</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>Exhaust Valves</td>
<td>1</td>
<td>1.5</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>Intake Valves</td>
<td>1</td>
<td>1.5</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>Cylinder Head</td>
<td>3</td>
<td>5</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>Water cooling Jacket</td>
<td>1.5</td>
<td>2</td>
<td>5 (ratio=1.2)</td>
<td>0.5</td>
</tr>
<tr>
<td>Cylinder block</td>
<td>3</td>
<td>5</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>Liner</td>
<td>2</td>
<td>3</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>Saver ring</td>
<td>1</td>
<td>1.5</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>Gasket</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Figure 19: Selected polyhedral mesh for the full model (complete 1-cylinder). Cut view through the cylinder head, block and liner. Blue cells indicate cells corresponding to the WCJ domain.
Fig. 20: Details of the inflation layer generated for the selected polyhedral mesh after the mesh sensitivity study. Water cooling jacket domain has 5 inflation layers with a total thickness of 0.5 mm (Blue) and solid domains have 1 inflation layer of total thickness of 0.5 mm (Green).

### 3.2.2 Setup and simulation

Similarly as in the first stage, material information is assigned to all the domains, including the new parts (cylinder block, liner, gasket and saver ring). As in the first stage, material properties are temperature dependent and are loaded into the model. Materials corresponding to all parts or domains of the 1-cylinder model of the studied engine are shown in Table 13.

**Tab. 13:** Materials corresponding to each engine component during the setup for the complete Scania geometry. These materials have temperature dependent properties specified in the AVL Fire software via global formula (See Appendix).

<table>
<thead>
<tr>
<th>Part</th>
<th>Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>Exhaust Seats</td>
<td>AR20 (Steel alloy)</td>
</tr>
<tr>
<td>Intake Seats</td>
<td>PL12 (Steel alloy)</td>
</tr>
<tr>
<td>Guides</td>
<td>GJL300 (Cast Iron)</td>
</tr>
<tr>
<td>Exhaust Valves</td>
<td>X50CrMnNiNbN21 (Stainless steel – heat treatment)</td>
</tr>
<tr>
<td>Intake Valves</td>
<td>Nimonic (nickel-based high-temperature low creep super alloys)</td>
</tr>
<tr>
<td>Cylinder Head</td>
<td>GJL300 (Cast Iron)</td>
</tr>
<tr>
<td>Water cooling Jacket</td>
<td>Water 60%/Glycol 40% (Engine coolant)</td>
</tr>
<tr>
<td>Cylinder Block</td>
<td>GJV450 (Cast Iron)</td>
</tr>
<tr>
<td>Liner</td>
<td>GJL300 (Cast Iron)</td>
</tr>
<tr>
<td>Saver ring</td>
<td>GJL300 (Cast Iron)</td>
</tr>
<tr>
<td>Gasket</td>
<td>GJL300 (Cast Iron)</td>
</tr>
</tbody>
</table>
Coolant conditions and properties remain the same as the ones shown in Table 7 in the Setup and Simulation section from the previous stage.

An important modification from the first stage, corresponds to the addition of more realistic boundary conditions, this being referred to boundary conditions of thermal parameters like temperature and heat transfer coefficients, directly mapped from combustion simulations without space averaging. These boundary conditions correspond to time average values of the engine cycle simulated for the combustion process.

Recommendations from AVL Fire support are to proceed with the mapping of the boundary conditions using the AVL Fire classic version of the software, instead of the AVL Fire M version, since this feature is not directly available in the version of the AVL Fire M version used (v.2017).

From the previous recommendations, several changes had to be made in the implemented methodology used for the simplified geometry study, important amount of effort was dedicated to the implementations and modelling of a multi-material project in the AVL Fire classic version, where the procedure is different, since there is the need of using “dummy files” and global formulas (scripts) to add the different material properties as shown in the Appendix. This new approach in AVL Fire classic, coming from a polyhedral mesh generated in AVL Fire M, was investigated during this work performing several tests within the company (Scania CV AB – NMGD Department).

After the multi-material approach in AVL Fire classic is defined and proved to be working, it is possible to “map” the boundary conditions from previous combustion simulations, the procedure to follow is similar to the one already implemented at Scania CV AB with the ACCI methodology, which is one of the advantages to migrate from AVL Fire M to AVL Fire classic after the generation of the polyhedral mesh.

Figures 21 shows the mapped values as boundary conditions for heat transfer coefficient and temperatures in the intake and exhaust ports, the valves and the fire deck.

**Fig. 21:** Mapped boundary conditions from combustion simulations in the intake and exhaust ports, and fire deck. Bottom view. Left: Heat Transfer Coefficient (HTC) values [W/m²K]. Right: Temperature [K]. Engine operating point of 1800 rpm with 115% load.

### 3.2.3 Efficient cluster use Study

An important topic to study during this project is to discover the best way to use the simulation resources within the company, in order to have an efficient use of the computer cores in the cluster and still achieve fast simulation times.

In order to do this, is it proposed to run the same model with different number of cores each time, this will provide the behavior of the cluster resources in terms of time, compared against the numbers of cores used. When the amount of time reduced by increasing the number cores per simulation is no longer relevant it is possible to say that it has been achieved an efficient simulation configuration between number of cores being used and the amount of polyhedral grid cells in our model, that will be valid for similar thermal studies of engine components and water cooling jackets using similar configurations and polyhedral meshes.

Another fact to be tested, is the recommendation given by AVL Fire support regarding the cells in the water cooling jacket. These cells demand more computational resources in order to solve all the physics involved as explained in the theory, Section 2.2, like velocity profiles,
pressure distribution, heat convection and boiling; compared against solid domains cells where only heat conduction is computed. The recommendation given is to specify a different computational weight for these specific cells in the water cooling jacket domain, by changing the name selection for this cells in a specific format (selectionname_decomp_weight_< >).

In this case, the water cooling jacket domain cells are set to have a computational weight ten times higher than the cells corresponding to solid domains, and its distribution along the different cluster cores will be even, improving the efficiency of the simulation (selectionname_decomp_weight_10).

A graph with the speed-up of computational times for different amount of computer cores and different cell weight decompositions is shown in Figure 22. The mesh used for this study corresponds to the base mesh that is also used as the starting point for the mesh sensitivity analysis.

![Fig. 22: Speed-up for different computational cores, 19 cores is the reference point. Base mesh from mesh sensitivity study is used (3.5 million cells). Blue curve represents simulations with same cell weights for all the domains, red curve corresponds to simulations with WCJ cell weights 10 times higher than solid domains cells, gray curve shows optimal speed-up line.](image)

From Figure 22 it is possible to say that, for the specific mesh used, there is a considerable time reduction, or an increment in the simulation speed, up to 95 cores, where the speed-up for the equal weight decomposition case is above 3, and for the different weight decomposition case regarding WCJ cells is close to 4 (this corresponds to a simulation time close to 8 hours). For a higher number of cores used in the simulations there is no relevant time reduction or speed-up, so it is not worth it to use an excess of computational resources.

When comparing the different weight decomposition simulations, it can be seen an increment of about 22% of the simulation speed due to the different cell weight definition assigned to the water cooling jacket domain.

From the cluster efficiency study it is possible to conclude that the most efficient use of the resources within thermal simulations of engine components, with similar settings as the ones used in the present project for polyhedral meshes, is to use a ratio about 40000 cells per computational core (3.5 million cells with 95 cores), together with the variable weight decomposition for cells in the fluid domain (WCJ cells 10 times the weight of solid domains cells).

### 3.2.4 Comparison study with experimental results

To validate the methodology, it is important to be able to compare the results with experimental tests that have been performed for similar cases and conditions. In order to do such a comparison, engine test results were gathered from the engine test department within Scania CV AB.
It was not possible to obtain experimental data for the same engine configuration that was used and modeled in the previous steps of the project (1800 rpm with 115%), since the data available corresponds to an engine configuration with a slightly different liner component, and for a different operating condition of 1900 rpm with a 100% of the load [14].

To validate the methodology against the experimental data for the new engine operating condition, the corresponding changes were implemented in the setup and simulation sections of this stage (Sections 3.2.1 and 3.2.2). The most relevant changes for this comparison correspond to a change in the boundary conditions mapped from combustion results and for the condition of the water cooling jacket, as it can be seen in Table 14, due to changes in the engine operating regime.

**Tab. 14:** Boundary conditions and properties for the coolant of the Water Cooling Jacket domain. Conditions correspond to engine operating point of 1900 rpm and 100% of load.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coolant glycol fraction</td>
<td>0.35</td>
</tr>
<tr>
<td>Inlet Boundary condition</td>
<td>Mass flow: 1.11 kg/s, Temp.: 77.45°C</td>
</tr>
<tr>
<td>Outlet Boundary condition</td>
<td>Static pressure: 2.91 bar</td>
</tr>
<tr>
<td>Deairation ports Boundary condition</td>
<td>Mass flow: -0.0068 kg/s</td>
</tr>
<tr>
<td>Density</td>
<td>1054.5 kg/m³</td>
</tr>
<tr>
<td>Reference temperature</td>
<td>293.15 K</td>
</tr>
</tbody>
</table>

Apart from being able to compare the results with experimental data, another advantage by using this engine configuration is that the former ACCI methodology results are available for this case, so it would be possible to compare the results against both experimental data and the former method simulation results.

The experimental data obtained correspond to measurements of metal temperature in 8 different locations of the cylinder head, in the region of the fire deck, at 1 mm from the surface of the metal. Positions of the temperature sensors are shown in Figure 23, these are the locations were temperature will be computed in the simulations to be able to directly compare the results, the center value of the closest cell in the grid to the specified point is the one used.

![Fig. 23: Temperature sensor locations (red dots) in the cylinder head for comparison between experimental and simulation data. Temperature measured at 1mm inside the metal surface.](image)
4. Results

4.1 Stage 1: Simplified SCANIA Geometry

In this section, results obtained for the simple Scania engine geometry, with the use of a polyhedral mesh and average boundary conditions from existing combustion simulations are shown and compared against result from the former methodology (ACCI method with hexahedral mesh), as explained in the Method section. It is important to comment that for the former method simulations, mapped boundary conditions are used, exactly from combustion analysis, so this comparison is only to validate the overall behavior of the new method and test its stability, it is not intended to compare exact fluid dynamics results.

*Fig. 24:* Boiling presence water cooling jacket simulation comparison. Left: Former methodology with hexahedral mesh (AVL Fire classic). Right: New methodology with polyhedral mesh (AVL Fire M). Overall boiling presence identified in the same regions for both simulations.

In Figure 24 is it shown the results for the single-phase boiling module, comparing the former method and the new proposed method. Similar boiling regions can be identified in both simulations, specifically in the hottest regions of the water cooling jacket next to the exhaust ports of the cylinder head. Partial agreements between the solutions validate the results from the new methodology with the implementation of the polyhedral mesh. This is an important step to fulfil in order to continue with the study an implementation of this new procedure for further simulations. Due to the different boundary conditions used for the new methodology, applying average values instead of mapped values directly from combustion simulations, the results are expected to differ.
**Fig. 25:** Cylinder head temperature distribution [°C]. Left: Former methodology with hexahedral mesh (AVL Fire classic). Right: New methodology with polyhedral mesh (AVL Fire-M). New method uses simplified boundary conditions. Maximum and minimum temperature values are found in the same regions for both simulations. Temperature values are hidden for confidentiality reasons at Scania CV AB. Same temperature scale is used for both cases shown.

Figure 25 presents a comparison between temperature distribution obtained with the former methodology and the new one, it is clear that by using averaged boundary conditions in the simplified model less amount of energy is given as an input, obtaining considerable lower temperatures in the different components and regions of the cylinder head. In Figure 25 it can be seen that the temperature distribution along the cylinder head and different components like intake and exhaust valves show similarities, maintaining the same pattern for both simulations as expected, providing another sign of validation of the new methodology.

**Fig. 26:** Thermal contact resistance effects on temperature distribution between material interfaces [°C]. Left: Thermal contact resistance between exhaust valves and seats set to 1e-4 [m²K/W] (Ability to define it in the new method). Right: Thermal contact resistance between exhaust valves and seat set to zero (0) [m²K/W-1] (Only option in former method). Temperature values are hidden for confidentiality reasons at Scania CV AB. Same temperature scale is used for both cases shown.

Figure 26 shows an important difference on the temperatures around the contact region between exhaust valves and exhaust seats. In the new method, it is possible to introduce thermal contact resistance values to model the periodical contact between these components,
due to the normal engine cycle, adding a recommended value from AVL Fire support of $1e^{-4}$ [m$^2$K/W] in these domains interfaces.

From Figure 26, it is clear that in the case of thermal contact resistance set to zero, which is the only available setting in the former method, the temperature distribution is continuous, producing higher temperatures in the seats contact regions but lower temperatures for the valves contact regions. In the case of the recommended value for thermal contact resistance, it can be seen an abrupt change in the temperature distribution for the contact region, with differences around 100-200 °C, generating higher temperatures in the valves interfaces and lower temperatures seats interfaces. It is important to mention that differences in the results when applying thermal contact resistance values, does not affect the rest of the components of the engine, regarding temperature profiles, and heat flux to the coolant. In this case, the impact is negligible for the rest of the model since the contact regions have a small area, having insignificant effect in terms of heat transfer.

### 4.2 Stage 2: Complete SCANIA Geometry (1-cylinder)

This section shows the results from the complete Scania geometry simulations with the new methodology, and a comparison with the results from the former method. In this case, boundary conditions are directly mapped from combustion simulations at an engine operating point of 1800 rpm and 115% load. Different from the first stage, in this section it is of interest to compare the exact results from each case, considering all the fluid variables like pressure and temperatures in the coolant, as well as metal temperatures in the engine components.

Table 15 shows important thermal parameters obtained for the different simulations performed. Different cases shown correspond to the former methodology, the new methodology with same inflation layer settings used in the former method (1 layer of 0.5 mm thickness), and the new methodology with the mesh selected from the sensitivity analysis which has low $y^+$ values (5 inflation layers with a 0.5mm total thickness).

**Tab. 15**: Thermal simulations results. Methodology comparison. All values have been normalized with the Former ACCI Method results used as reference.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Former ACCI Method</td>
<td>1.000</td>
<td>1.000</td>
<td>1.000</td>
<td>1.000</td>
<td>1.000</td>
<td>1.000</td>
<td>1.000</td>
</tr>
<tr>
<td>(Hexa-mesh high $y^+$)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>New Method</td>
<td>0.885</td>
<td>0.929</td>
<td>0.718</td>
<td>0.977</td>
<td>0.925</td>
<td>0.853</td>
<td>0.987</td>
</tr>
<tr>
<td>Base mesh (Poly-mesh high $y^+$)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>New Method</td>
<td>0.943</td>
<td>1.089</td>
<td>0.374</td>
<td>0.549</td>
<td>0.942</td>
<td>0.973</td>
<td>0.996</td>
</tr>
<tr>
<td>Selected mesh (Poly mesh low $y^+$)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The results shown in Table 15 present an overall good agreement when comparing the former methodology with the new proposed method using similar meshing settings, results for the studied thermal parameters have similar values, with a difference of 12% in the total heat transferred, computed for both methods.

Similarities can also be seen in the boiling parameters between the two methodologies using same mesh inflation settings, having similar boiling areas, and similar boiling heat ratios when
compared to the total heat, with values of 20% for the former method and 17% for the new methodology simulation.

When comparing the results for the new methodology with the selected mesh from the sensitivity analysis, similarities are found in the total heat transferred, but important differences can be seen in the boiling parameters, with a relevant reduction of 43% in the boiling area when compared to the initial mesh settings results, as it can also be seen in Figure 27.

Computation of the maximum temperature for the cylinder head with both methodologies and using different mesh arrangements provides similar results in all the cases, with differences within an acceptable range of 5 degrees.

![Boiling Presence](image)

Fig. 27: Boiling presence in the engine water cooling jacket. Results from former and new proposed methodology. Similarities can be seen between the two methodologies for the same mesh settings regarding inflation layers. Results obtained with new method and mesh selected from the mesh sensitivity analysis shows a reduced boiling presence, and it is presumed to be more accurate.

Similarities are also shown in Figure 27 where the boiling presence for the two methodologies with same meshing settings is practically equal, identifying boiling regions in the same areas of the water cooling jacket.

An important reduction of the boiling presence is identified for the new methodology simulation with the selected low $y^+$ mesh, concentrating the boiling region around the hottest section in the cylinder head like the exhaust ports.
Fig. 28: Temperature distribution in the water cooling jacket walls. Former methodology results show similarities with the new method for similar mesh settings. More accurate results can be seen in the new method using the selected mesh, with concentration of higher temperatures in the water cooling jacket walls next to the exhaust ports of the cylinder head. Temperature values are hidden for confidentiality reasons at Scania CV AB. Same temperature scale is used for all cases shown.

Figure 28 provides a more accurate temperature distribution in the water cooling jacket walls for the new methodology results with the low $y^+$ grid selected from the mesh sensitivity study, than the simulations with meshes having higher $y^+$ values. With the low $y^+$ mesh, regions of high temperatures above the saturation point of the coolant are covering less area, but with higher maximum temperature values concentrated in certain regions of the water cooling jacket walls, specifically next to the exhaust ports of the cylinder head, which corresponds to one of the hottest areas of this component.

On the other hand, for the meshes with higher $y^+$ values, temperature is more evenly distributed along the water cooling jacket walls, having larger regions with temperatures above the saturation point of the coolant, but being unable to identify high wall temperature concentration spots, with maximum values considerable lower than the ones found in the mesh independent simulation.
Fig. 29: Temperature distribution in cylinder head. Overall, the results show similar behavior for both the former and new methodology, as well as for the low $y^+$ mesh settings case. Only the surface temperature is shown and this is highly affected by the boundary conditions, which are the same for all the cases. Temperature values are hidden for confidentiality reasons at Scania CV AB. Same temperature scale is used for all cases shown.

Figure 29 shows temperature distribution along the exterior surfaces of the cylinder head, and results are similar for all the simulations compared. Having such similar results from both methodologies is another proof to validate the new proposed method against the former methodology for these types of simulations. Temperature distribution, together with concentration of maximum temperatures can be seen in the same regions of the cylinder head, specifically on the fire deck.

To continue the analysis regarding the temperatures obtained in the cylinder head between different methodologies, it is of interest to study the results in each one of the locations of the temperature sensors used for engine experimental tests.

### 4.2.1 Comparison with experimental results

As mentioned in Section 3.2.4, results from experimental test available correspond to a different engine with changes in the liner component, and running at a different operating regime, at 1900 rpm with 100% load.

Temperature values at different sensor locations are compared between the different methodologies against the experimental test results. The data gathered from the experimental test does not provide results for the sensor location number one (#1) due to errors in the measurements.
Tab. 16: Temperature difference between simulation methods when compared to experimental test measurements. Engine operating point: 1900 rpm 100% load. Green: closer result to engine test; Red: Farthest result from engine test. Test measurements values are hidden for confidentiality reasons at Scania CV AB.

<table>
<thead>
<tr>
<th>Sensor #</th>
<th>Test Measurements</th>
<th>Former ACCI method</th>
<th>New multi-material method</th>
</tr>
</thead>
<tbody>
<tr>
<td>#2</td>
<td>-21.3</td>
<td>-11.3</td>
<td>47.6</td>
</tr>
<tr>
<td>#3</td>
<td>-23.6</td>
<td>17.3</td>
<td>-18.4</td>
</tr>
<tr>
<td>#4</td>
<td>-19.6</td>
<td>-7.1</td>
<td>22.7</td>
</tr>
<tr>
<td>#5</td>
<td>-20.5</td>
<td>-6.6</td>
<td></td>
</tr>
<tr>
<td>#6</td>
<td>-8.6</td>
<td>47.6</td>
<td></td>
</tr>
<tr>
<td>#7</td>
<td>-21.0</td>
<td>-18.4</td>
<td></td>
</tr>
<tr>
<td>#8</td>
<td>-2.8</td>
<td>22.7</td>
<td></td>
</tr>
</tbody>
</table>

Table 16 shows that sensor temperatures computed with the new multi-material methodology present a better agreement with experimental or tests results in locations #2, #3, #4, #5, and #7, while locations #6 and #8 show an increment in the prediction of the local temperature. In five out seven available sensor locations the results are closer to reality with the new methodology.

Fig. 30: Temperature at sensor locations. Comparison between test measurements and the results from the Former and the New Simulation methodologies. Results obtained for the New methodology are closer to experimental results in five out of seven sensor locations. Engine operating point: 1900 rpm 100% load. Test measurements values are hidden for confidentiality reasons at Scania CV AB.

Figure 30 shows that the temperature distribution profiles and the patterns obtained with the new multi-material methodology agrees with the test results analyzed, following the same trend and identifying correctly the hottest and coldest points in the cylinder head.
5. Discussion

5.1 Stage 1: Simplified SCANIA Geometry

From the results for the simulations with the simplified Scania Geometry, similarities can be identified between the former and the new methodology for thermal simulations of engine components.

Due to the simplifications applied in the model to test the new method, it is expected to obtain different results when comparing parameters one to one. These simplifications include a crucial one like the space averaging per component of the boundary conditions obtained from combustion simulations results. By implementing the simplifications, less amount of energy is given as an input to the system, being the reason for a lower temperature profile along the entire model, as well for a reduction in the region with boiling presence, as it can be seen in Figures 24 and 25.

The main objective for these simplified simulations was to test the new methodology in a fast and efficient way, to be able to identify problems or issues regarding the method in an early stage of the project. During this stage, it was possible to test the polyhedral mesh generation tool to generate a multi-domain grid, automatically coupled, since there was no previous experience within the department regarding this type of meshes.

After analyzing the results obtained with the new method, and comparing them with the former method it is possible to identify, in Figures 24 and 25, similarities in the boiling and temperature patterns along the model, since boiling regions correspond to similar areas in both simulations and temperature distribution shows equal behavior. From these results, the new methodology is proved to be working in a satisfactory way when compared to the former method, being a good starting point for the project and the next stages of it.

Important differences are identified between the former and the new methodology in Figure 26, with the implementation of a thermal contact resistance for periodical interfaces between valves and seats. This may represent an important factor when moving forward to evaluate thermo-mechanical stresses for these engine components, since temperature differences between these interfaces can reach values around 100-200 Δ˚C, and this effect was not capture with the former method where temperature distribution is continuous and both interfaces show the same temperature values.

The use of thermal contact resistance in these areas has a negligible effect on the rest of the model due to the reduced transfer area. However, the implementation of thermal contact resistance values in regions of periodical contact, represents a more realistic configuration or setup, as long as the thermal resistance value implemented is accurate enough to model the cyclic contact correctly.

5.2 Stage 2: Complete SCANIA Geometry (1-cylinder)

For the second stage of the project, simulations were performed without any simplifications when compared to the former method as explained in Section 3, and reproducing the same type of setup that is used within the NMGD department at Scania CV AB.

Agreement in the results between different methods fully validate the new methodology proposed, since for similar mesh setting to the ones used for the former methodology, results are practically the same and reliable. Nevertheless, it is important to mention that this mesh configuration is not mesh dependent according to the sensitivity analysis performed in the Method section, and this is the reason for running simulations with the mesh selected from the sensitivity analysis, having the characteristic of low $y^+$ values in the fluid domain inside the
water cooling jacket ($y^+$ values close to 1). No mesh sensitivity study exists for simulations using the former ACCI methodology.

The differences in the boiling area computations shown in Table 15 and Figure 27 can be explained analyzing the results for the temperature profile in the water cooling jacket walls, shown in Figure 28, specifically for the mesh with low $y^+$ values close to 1. The simulation on this mesh captures a reduction in the areas with temperatures above the saturation temperatures of the coolant, but with higher temperature values concentrated in the region next to the exhaust ports of the cylinder head where the hottest combustion gases can be found.

From this analysis, it can be said that the hybrid-wall treatment used in the simulations for solving meshes with $y^+$ values higher than 5, is able to capture the global behavior of the problem and its physics, but penalizing the accuracy in terms of temperature distribution along the walls of the water cooling jacket, also affecting the presence of boiling regions and the boiling heat contribution.

By a reduction in the $y^+$ values, aiming for numbers close to one (1) on the water cooling jacket domain, fluid equations are solved and integrated up to the wall, as explained in the theory section, instead of the use of blending functions or wall functions. This is possible due to the advantage of the $\tilde{\zeta}$-$f$ turbulence model used and its capability to be integrated up to the wall, making the solution mesh independent and significantly accurate and reliable when compared to experimental results discussed in Section 5.2.1. More explanation regarding the impact of the inflation layer settings and the differences in $y^+$ values for the grid are pointed out in the mesh sensitivity study in the method Section 3.2.1 of this report.

It is important to mention that in these types of problems it is possible to generate polyhedral grids with low $y^+$ values close to 1 in the water cooling jacket domain, since the Reynolds number is considered to be low within the turbulent range as shown in Section 3, making these options viable in terms of computational resources within the working department at Scania CV AB. Having a low Reynolds number allows thicker cells in the inflation layer to obtain low $y^+$ values, considerably reducing the number of cells needed in the inflation layer, and reducing the risk of bad quality cells in the wall region.

Similarities are found in Figure 29 for the temperature distribution in the surfaces of the cylinder head. One reason that explains the equality of the results for the surface temperatures of this component, is that the same convective boundary conditions are being used for all the cases, mapped directly from the same combustion simulations, so it is expected to have also similar temperature results in the corresponding surfaces where these boundary conditions are being applied.

To prove that results obtained for the new methodology with the selected mesh of low $y^+$ values are the most accurate ones, they are compared with experimental tests data, by measuring temperatures in different sensor locations on the cylinder head, discussion regarding this comparison in carried out in the Section 5.2.1.

### 5.2.1 Comparison with experimental results

After analyzing the results comparing the measured sensor temperatures with results from both the former ACCI method and the new method, the new multi-material approach with the selected mesh from the mesh sensitivity study provides a more accurate temperature prediction to experimental data in five out of seven of the measurement points, as it can be seen in Table 16 and Figure 30.

In location #6 there is an over-estimation of the temperature value when using the new method in comparison with experimental data, however, it is important to mention that at this location is where the highest temperatures of the cylinder head will be generally found for all engine operating conditions, as it can be seen in Figure 31. This point it’s close to the center of the combustion chamber and in between the two exhaust ports. In this critical location there are also large temperature gradients within very small distances, and since the temperature is computed in the closest cell to the desired point, its center may be deviated by 1.5 mm, which is the refinement level of the cells in this region according the method section, providing the
temperature value of a slightly different location in the cylinder head and reflecting significant differences.

It is curious to note that the difference for this point (#6) between the former method and the test measurement is smaller than the tendency observed for the other locations, where computed temperatures are lower than the measurements keeping a similar range of degrees below. The value obtained in the former method for point #6 is changing the pattern according to previous sensor locations and makes it hard to identify it as a reliable result or as a wrong measurement of the temperature at that position.

Differences between measurements of point #6 and the new method could be attributed to errors in the provided measured data, by placing the sensor at a slightly different position than supposed or due to misleading reading of the temperature during the test, since sensor #6 is relatively close to the location of sensor #1, which had errors and was not able to provide a correct lecture of the temperature at that location. Similar concerns are identified for sensor location #8, where the experimental value measured is similar to the one in #4, but logically the temperature in #8 should be higher than #4 due to being a point closer to the center of the combustion chamber (Figure 25) where temperatures in general are higher; the trend of both former and new simulations show the expected behavior, having higher temperatures for location #8 than for location #4.

Comparison with experimental results fully validates the implementation of the new multi-material methodology, that additionally to the benefits obtained in the simulation process is able to provide reliable and satisfactory results in agreement with engine test data.

5.2.2 Engineering and Computational resources

After analyzing the Method presented and tested in this report, together with the results obtained, it is proven that the new methodology simulation can be performed by using only one software license for all the model, instead of one license per domain that is needed in the former method due to the parallelization of the simulation, as it is stated in the Introduction section of this work.

For the new methodology, the mesh is generated in one go, in a unique file, with automatic recognition of domain interfaces, simplifying considerably the setup of the simulation, making a significant difference against the former method where one mesh per domain is needed, and the identification of interfaces needs to be done in manual process consuming engineering resources and time.

The study of the use of the resources in the cluster presented in Section 3.2.3 shows that the best configuration for a fast simulation is to assign around 40000 polyhedral cells per computational core, using the weight decomposition for the water cooling jacket cells. This configuration gives a speed-up close to 4 with simulation times around 10 hours for a similar model with the proposed mesh settings, which is an acceptable time for this type of simulations within the Scania NMGD department.

Another advantage is identified in the post-processing stage since in the new method there is only one setup case for the complete model, instead of one mesh and case per each domain required in the former methodology. This makes the analysis of the results much easier, significantly reducing the effort and the number of steps to analyze all the model.
6. Conclusions

From the results and discussion presented in this study, it is to conclude that a new multi-material thermal simulation methodology, implementing a polyhedral type mesh, have been evaluated within the Scania NMGD engine development department, providing reliable and satisfactory results when compared to the former method and with experimental results from engine tests carried out at the company.

During this work it was possible to implement a CFD conjugate heat transfer methodology using the multi-material advantages of the software AVL Fire M and AVL Fire classic, generating the polyhedral mesh with the mesh tool from AVL Fire M, and proceeding with the setup, simulation and post-processing within AVL Fire classic.

Difficulties were encountered when migrating from the M to classic version of AVL Fire, due to the complexity added when running multi-material projects in the classic version of the software since there was no previous experience with these types of simulations at the department. However, this step is necessary in order to be able to map boundary conditions directly from earlier combustion simulations. The mentioned compatibility difficulties were solved after several discussions with AVL Fire support personnel, proposed procedures were applied and integrated in the methodology report delivered to Scania NMGD department.

The polyhedral mesh type used for this project provides a robust solution to generate a grid in complex geometries like water cooling jackets, no major refinements are needed and quality of the obtained mesh passed orthogonality, skewness, and other mesh requirements like negative volume cells, in contrast to previous experience with hexahedral meshes from the former methodology which are much more sensible in terms of the mentioned quality parameters, giving problems in the generated cells for these complex geometries or domains.

The implementation of thermal contact resistance values between domain interfaces that in the real case have a periodic contact, like between valves and seats, allows to simulate this periodic contact within steady state simulations. This possibility is a new feature of the proposed multi-material simulation, providing more realistic results with an important impact in the temperature distribution at the interfaces, where temperatures in the interface of one component show differences around 100-200°C against the same interface in the other component. These results do not affect the rest of the temperatures profiles of other engine components due to the small transfer area, but may be important when performing thermomechanical analysis of the affected parts (valves, seats and guides).

An important finding of this study is the need of a mesh with low $y^+$ values in the water cooling jacket domain, to be able to capture the correct temperature distribution in the jacket walls and a precise boiling presence region. The mesh sensitivity analysis shows that the use of the hybrid wall treatment with high $y^+$ values can provide overall good results for the total heat transferred and temperature distribution of the solids away from the water cooling jacket, but in order to have an independent mesh in terms of boiling presence and temperatures around the jacket walls, integration of the $\epsilon-\mu$ turbulence model up to the wall is needed by the use of low $y^+$ values close to one (1).

Stability issues identified in the single-phase boiling module where significantly reduced by decreasing the under-relaxation factors for the energy equation, by the reduction of the $y^+$ values in the water cooling jacket of the simulation mesh, and following recommendations from AVL Fire support to activate the module from a partially converged solution, in this case after 2000 iterations.

In terms of efficiency improvements, the new method allows the use of only one software license to perform the simulations, simplification of the meshing process together with automatic identification of domain interfaces, simplification of the setup process allowing one unique case for the entire model, simulations can be performed within acceptable time for the required convergence criteria by using the recommendation of 40000 polyhedral cells per computational core, and simplification of the post-processing stage.
Overall, it can be said that the results from this project are satisfactory, fulfilling the objectives of the thesis presented in the Introduction chapter, providing the NMGD department of Scania CV AB with an improved, simplified, and robust methodology for CFD conjugate heat transfer multi-material simulations for engine components.
7. Outlook

Further work is suggested in this chapter aiming to improve the proposed methodology evaluated in this project, and to study the impact that this method may have in the development of new products at Scania CV AB.

A recommendation would be to investigate and compare simulations performed with the new methodology with recent experimental results for different engine configurations and operating regimes. Data gathered from experimental tests is relatively old, being from year 2009, so new measurements and validation will provide more reliable information, specifically in the hottest regions of the fire deck in the cylinder head, where high temperature gradients can be found for very small areas.

It is of interest to investigate the impact that the implementation of the thermal contact resistance values in periodical interfaces of valves, seats and guides, may have on the thermo-mechanical analysis of the affected components, since temperatures computed for the corresponding interfaces in the materials show important differences when compared to the previous approach where no thermal contact resistance values are being used.

If mapping of boundary conditions from combustion simulations is available and reliable to be used within future versions of the AVL Fire M software, it is of interest to investigate the implementation of the new methodology entirely in the Fire M tool, avoiding the migration to AVL Fire classic and simplifying the steps to setup a CFD thermal multi-material simulation for engine components.
8. Perspectives

This chapter discusses the effects of this project in societal and commercial perspectives such as environment, economy, society, working environment and ethics.

From an environment point of view, improvements of thermal simulations of internal combustion engines lead to the design of more efficient machines for the transport industry, contributing to the reduction of emissions of greenhouse gases into the atmosphere.

The new simulation method has an important economic impact within the company, since less computational resources like software licenses are needed to perform the required multi-material thermal simulations, reducing the costs associated to this task with respect to former methodology where several software licenses are required.

Findings of this thesis also have a positive impact in the engineering resources needed, since process is optimized and complexity is reduced for the engineer doing the work. This means that the Design CFD engineer can invest more time in the development and study of the product, analyzing results and testing several configurations of a model to improve efficiency, instead of doing repetitive tasks with no significant engineering value. In the same order of ideas, if the engineer feels comfortable and productive with the work done, the working environment is improved.

Results from this work are considered to have a small impact in social and ethical considerations.
References

Appendix

A.1 Name selections Figures and Tables (pre-processing stage)

**Fig. A1**: ANSA surface representation. Cylinder head with created named selections. View from below (not all selections shown in the figure).

**Fig. A2**: ANSA surface representation. Cylinder head with created named selections. View from the Exhaust Flange (not all selections shown in the figure).
Fig. A3: ANSA surface representation. Cylinder head with created named selections. Cut-view through the Exhaust Port (not all selections shown in the figure).

Fig. A4: ANSA surface representation. Cylinder block with name selections. View from above-rear side (not all selections shown in the figure).
Fig. A5: ANSA surface representation. Cylinder block with name selections. Cut-view through cylinder (not all selections shown in the figure).

Fig. A6: ANSA model 1-cylinder SCANIA engine geometry. Geometry checks have been performed on the geometry in order to have a good quality surface mesh for further preparations in Fire M (Triple Cons are allowed and necessary within the surface mesh).
**Tab. A1:** Face selections and descriptions. Example Table for the Cylinder Head Component.

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>INT_gasket_head</td>
<td>The surface facing and shared with the gasket</td>
<td>Define interface between domains</td>
</tr>
<tr>
<td>INT_head_exhaust_guide</td>
<td>The contact surface shared with the exhaust guides</td>
<td>Define interface between domains</td>
</tr>
<tr>
<td>INT_head_exhaust_seat</td>
<td>The contact surface shared with the exhaust seats</td>
<td>Define interface between domains</td>
</tr>
<tr>
<td>INT_head_intake_guide</td>
<td>The contact surface shared with the intake guides</td>
<td>Define interface between domains</td>
</tr>
<tr>
<td>INT_head_intake_seat</td>
<td>The contact surface shared with the intake seats</td>
<td>Define interface between domains</td>
</tr>
<tr>
<td>INT_wcj_head</td>
<td>The contact surface shared with the wcj</td>
<td>Define interface between domains</td>
</tr>
<tr>
<td>BND_air_gap</td>
<td></td>
<td>Define BC.</td>
</tr>
<tr>
<td>BND_bolt_hole</td>
<td></td>
<td>Define BC.</td>
</tr>
<tr>
<td>BND_exhaust_flange</td>
<td></td>
<td>Define BC.</td>
</tr>
<tr>
<td>BND_exhaust_port</td>
<td></td>
<td>Define BC. Mapped.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Refinement: cell size 1.5, refinement depth of 3 mm</td>
</tr>
<tr>
<td>BND_exterior</td>
<td></td>
<td>Define BC.</td>
</tr>
<tr>
<td>BND_fire_deck</td>
<td></td>
<td>Define BC. Mapped.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Refinement: cell size 1.5, refinement depth of 3 mm</td>
</tr>
<tr>
<td>BND_fuel_rail</td>
<td></td>
<td>Define BC.</td>
</tr>
<tr>
<td>BNDInjector_bottom</td>
<td></td>
<td>Define BC.</td>
</tr>
<tr>
<td>BND_intake_flange</td>
<td></td>
<td>Define BC.</td>
</tr>
<tr>
<td>BND_intake_port_curved</td>
<td></td>
<td>Define BC. Mapped.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Refinement: cell size 1.5, refinement depth of 3 mm</td>
</tr>
<tr>
<td>BND_intake_port_straight</td>
<td></td>
<td>Define BC. Mapped.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Refinement: cell size 1.5, refinement depth of 3 mm</td>
</tr>
<tr>
<td>BND_oil_passage</td>
<td></td>
<td>Define BC.</td>
</tr>
<tr>
<td>BND_saver_ring_top</td>
<td></td>
<td>Define BC.</td>
</tr>
<tr>
<td>BND_valve_cover_gasket</td>
<td></td>
<td>Define BC.</td>
</tr>
<tr>
<td>BND_valve_train</td>
<td></td>
<td>Define BC.</td>
</tr>
</tbody>
</table>
A.2 Setup and Simulation Multi-material files

The following *.dat and *.txt files must be saved within the case folder for multi-material projects with AVL Fire classic:

Multi-material dummy file content (file name: multimaterial.dat):

```
# Number of material selections
2
# Name of solids selection
Material2
# Density [kg/m3] (dummy, is overwritten by USEDEN) value
# Thermal conductivity [W/mK] (dummy is overwritten by USECON) value
# Heat capacity [J/kgK] (is formally used, but density is corrected) value
# Dynamic viscosity [Ns/m2] value
```

Temperature dependent thermal conductivity file content for different materials (first column: Temperature [K], second column: thermal conductivity [W m⁻¹ K⁻¹]):

Material X (file name: X_Conductivity.txt):

```
Temp_value_1  Conductivity_value_1
Temp_value_2  Conductivity_value_2
Temp_value_3  Conductivity_value_3
Temp_value_4  Conductivity_value_4
Temp_value_5  Conductivity_value_5
Temp_value_n  Conductivity_value_n
```