A conjugate heat transfer method for modeling oil-cooled pistons

CES Seminar

Julian Miller
Chair for Computational Analysis of Technical Systems,
Schinkelstrasse 2,
52062 Aachen, Germany
julian.miller@rwth-aachen.de

Abstract. In recent years, the power density of petrol engines has increased dramatically due to the need for lighter, more powerful engines with low emissions. To reach a higher power density, engine manufacturers use several methods to enhance the efficiency of engines which results in an increase in heat production. This raise in thermal loading of the piston leads to higher stresses on the engine components and may introduce seizure of the piston. To handle the improvement in power density, a sophisticated cooling of the piston is needed. Most commonly, pistons are cooled by oil jet impingement from the underside of the piston.

This paper provides a model of an oil-cooled piston based on a literature review of similar problems. The aim of the model is to enable the simulation of the exchange of thermal energy inside a piston. In the context of an oil-cooled piston, the exchange of thermal energy occurs between the solid piston and the cooling oil. This exchange is an interaction between the solid and fluid parts of the model. Problems containing such an interaction are called fluid-structure interaction (FSI) problems. To numerically solve the exchange of thermal energy, various methods are evaluated and compared. Based on this comparison, a conjugate heat transfer (CHT) method [20,16] is proposed due to its applicability to this problem and its modularity. The proposed conjugate method could be utilized for testing a wide range of piston designs which may be unsuitable for prototyping due to technical or economical difficulties. This allows for changes to the design of the piston to be evaluated in regards to their impact on the temperature distribution of the piston. Additionally, these computational tests can detect areas of great stress even before prototyping. Furthermore, the cooling could be optimized by simulating variations in speed and opening angle of the oil jet, different positions of the nozzle and different oil types. Thus a sophisticated model provides a quicker and, therefore, cheaper development process as well as a decrease in time-to-market for engine manufacturers.

Keywords: fluid-structure interaction, piston design, conjugate heat transfer, oil jet cooling, volume of fluid method, level set method, parallelization
1 Preliminaries

Prior to modeling the problem of an oil-cooled piston, a literature review was carried out. This section provides the results of this survey which covers the background of general FSI and, in particular, for piston cooling applications.

1.1 Fluid-structure interaction (FSI) problems

Fluid-structure interaction (FSI) describes the interaction between a solid and a liquid phase. In this case, the solid and liquid phases involve the piston itself and the flow of cooling oil and air underneath the piston respectively. FSI problems are multi-physics problems with applications in many scientific and engineering fields. Nevertheless, comprehensive solutions to FSI problems remain a great challenge in the field of computational fluid dynamics (CFD) due to the interdisciplinary nature of the problems and the resulting nonlinearity of FSI models [7,11,18]. FSI problems are typically impossible to solve analytically due to complex geometries which may also be deformable [21]. These deformations of the geometry lead to large computational loads due to the frequent remeshing. Apart from this, the interactive nature of these problems results in changes to the overall constraints at every single time step since the solid and fluid problem are coupled. Similar, experimental setups are limited in scope. Thus, numerical simulations are typically used to solve FSI problems.

The numerical methods of solving FSI problems are classified into monolithic and partitioned approaches. Regarding a monolithic approach, all governing equations of the fluid and solid problem form a single system of equations and are solved simultaneously with a single solver [19]. This unified mathematical framework consists of all governing equations and, implicitly, of the interfacial conditions. In general, monolithic code is developed for a particular problem and has to be reassessed for changes to the problem. This optimized approach for solving a single problem can achieve highly accurate solutions for FSI problems. Yet, developing a mathematical formulation for each particular problem may require more resources. This mathematical formulation will have to take extra care of the non-linearities arising from both physical domains. Solving strategies include legacy sparse matrix solvers.

Oppositely to the monolithic approach, the partitioned approach treats the solid and the fluid problem as two separate domains. Where the monolithic approach describes both domains using a single system of equations, this method uses multiple ones to optimize the mesh discretization but also the choice of solvers. Each of these domains consists of their respective grid and solving algorithm. Typical solvers are legacy codes for each domain which are extensively tested, validated and highly optimized in terms of execution speed and computational resources. Contrary to the monolithic approach, none of the domains of the partitioned approach carry interfacial conditions. Instead, these conditions are used to explicitly communicate between domains. This exchange of boundary
A conjugate heat transfer method for modeling oil-cooled pistons

conditions is reduced to the bare minimum. Therefore, the aim of the partitioned approach is to couple both domains as loosely as possible while maintaining the accuracy of the model. Depending on the target of the simulation, the coupling can be reduced to the exchange of only a few or even a single scalar variable between domains. However, the communication between disciplinary domains needs to be well-coordinated to achieve accurate results. Moreover, the explicit coupling of the partitioned approach can raise problems of numerical convergence. These problems restrict the choice of time steps of each iteration of both solvers and can be maintained by a problem-oriented synchronization strategy (see section 5.2).

In conclusion, loosely coupling of the domains typically leads to greater flexibility and modularity than strong coupling (monolithic approaches). Thus, the programming effort is lower for loosely coupled algorithms than for strongly coupled algorithms. Nevertheless, monolithic approaches tend to be more stable in their numerical solving process than partitioned approaches.

1.2 General piston theory

A piston denotes the moving component inside an internal combustion engine and it is guided in its movement by a liner. The top part of the piston called the piston crown transfers the force resulting from the combustion of the gases to the rotating crankshaft. The lower part of the piston is called undercrown. This area is usually used for oil cooling. Typically the ring belt consists of three rings which seal the combustion chamber. The pin boss is designed to hold the piston pin which connects the piston with the crankshaft. The area below the pin boss is called skirt. Figure 1.2 shows a scheme of a piston with the terminology defined above.

Fig. 1. Piston terminology[17]
1.3 Heat transfer

This paper focuses on the thermal analysis of pistons. In general, heat transfer describes the thermal energy that can be transferred through conduction, convection or radiation. Thermal conduction describes the transfer of kinetic and potential energy inside a body with a temperature gradient. The internal energy is a result of activity of molecules, atoms, electrons and phonons inside the body. Fourier’s law characterizes the transfer of internal energy as:

$$q = -\kappa \nabla T,$$

where $q$ denotes the heat flux, $\kappa$ the heat conductivity of the considered material and $\nabla T$ the temperature gradient. In a piston, thermal conduction occurs in the solid parts only.

On the other hand, convection occurs within fluids due to the collective movement of molecules. Convective heat transfer takes place by advection and diffusion and can be categorized by natural or forced convection. In the case of a piston, forced convection is used to cool the piston. This can be done by inserting a cooler fluid through a jet aiming at the hot bottom of the piston. Typical choices for the cooling fluid include water and oil while this paper focuses on oil jet cooling due to its greater cooling efficiency. The transfer of heat by convection is described by:

$$q = hA \nabla T,$$

where $q$ is the heat flux, $h$ the heat transfer coefficient, $A$ the heat transfer area of the surface and $\nabla T$ the temperature gradient. The temperature gradient is $\nabla T = T_s - T_f$, with the surface temperature of the solid object $T_s$ and the temperature of the fluid $T_f$. The heat transfer coefficient $h$ was measured and tabulated for various fluid flows. In the particular case of an oil jet, investigations of $h$ were carried out by Stevens and Webb in 1991 [22].

The third kind of heat transfer is radiation. Radiation or electromagnetic radiation is emitted by interatomic collisions in matter and occurs within all matter with a temperature above absolute zero. The exchange of thermal energy due to radiation is defined by the following equation:

$$\alpha + \rho + \tau = 1,$$

where $\alpha$ represents the spectral absorption component, $\rho$ the spectral reflection component and $\tau$ the spectral transmission component. Each of these components depends on the wavelength $\lambda$ of the electromagnetic radiation.

2 Model of the piston

Taking into consideration the literature survey presented above, a model of the piston was derived. This section provides an overview over the main parts considered by the model. These are the combustion chamber which acts as a heat
source on the top surface of the piston, the solid piston and the air or oil filled inner cooling gallery. The following section will elaborate on each part of the model in detail. After that, governing equations of both domains are introduced and two possible methods for solving the movement of the piston are disclosed.

The chemical reactions of the combustion can be seen as a source of heat which is applied to the top surface of the piston. In general, the combustion can be modeled with an Extended Coherent Flamelet Model (ECFM) [9] which features a two-step chemical reaction. An ECF model can be discretized with FEM which results in heat fluxes through the top surface of the piston. An alternative method are Large Eddy Simulations (LES) as proposed in e.g. [12]. The computed fluxes can then be used for a dynamical thermal simulation of the piston. However, since the piston is moving very fast the heat fluxes is assumed to be constant in time. Furthermore, the hot gases are presumed to spread out over the whole piston crown in a way that the heat flux on the piston crown is uniform over the surface. Taking these assumptions into account, the heat transfer of the piston can reach steady conditions if the constant heat source on top of the piston is balanced by the cooling jet underneath. Steady conditions present the most valuable information out of such a model from the standpoint of a manufacturer since the temperature of the body of the piston is usually the highest at steady conditions. Thus, the heating of the piston at engine start is not considered in this model. Note, that steady conditions can only be met in a single position of the piston. Hence, the simulation has to be carried out with a fixed positioning of the piston throughout the simulation. To cover the whole movement range of the piston the simulation has to be executed several times while varying the position of the piston.

A piston is mainly constituted of metal alloy. In the following it is assumed that the heat conductivity of the material is constant throughout the simulation. Moreover it is assumed, that the deformation of the piston due to the temperature is small compared to the size of the whole system. The surrounding cylinder is consequently assumed to be at constant temperature as well. The effect of thermal radiation of the body and its surroundings is then neglected. The heat exchange between the solid and fluid domain takes place at the interface of both domains. Therefore, the heat transfer coefficient $h$ underneath the piston has to be computed. $h$ is thereby influenced by various factors: oil flow, speed of the engine, oil nozzle diameter and angle, etc. To compute $h$ the predictive model of Stevens and Webb [22] may be applied.

The area under the piston is filled with air at atmospheric pressure. The fluid domain is bounded to the top and all sides by the inner walls of the piston. These are the undercrown and the inner part of the skirt. The lower border of the domain is introduced artificially connecting the bottom of the skirt. This border is set to be insulating due to the assumption of steady thermal conditions. Cooling oil is injected to this area through a jet nozzle. Therefore, the fluid
part has to be studied from a multiphase point of view where air or oil posses different parameters especially heat transfer properties. To keep track of the oil a free-surface modeling (FSM) method may be used. The most common FSM methods are the volume of fluid (VOF) method and level set methods (LSM). The VOF method is based on a volume fraction function $C$ with $0 \leq C \leq 1$ which represents the distribution of oil and air in each computational grid cell [15,14]. However, VOF codes are generally used for FVM only and do not provide sharp boundaries due to numerical diffusion at the interface. The LSM on the other hand provides a sharp boundary of the fluid-fluid interface. This method is great for shapes that undergo large topological changes like the moving piston but it lacks conservation of mass. In recent years, Bourlioux [6] and Sussman and Puckett [23] have combined the best of each of the methods to overcome the drawbacks of both methods. Nevertheless, these FSM methods only keep track of the fluid-fluid surface. The actual motion of the flow has to be solved separately which is governed by the Navier-Stokes equations.

2.1 Governing equations

The governing equations of the model can be split into three main parts: the governing equations of the solid domain, the governing equations of the fluid domain and boundary conditions.

Solid domain
Since the solid domain is assumed to have heat exchange through mainly conduction only the Fourier’s law has to apply. This main law of conduction can be solved with the steady-state Laplace-equation $\nabla^2 T = 0$.

Fluid domain
The main governing equations for the fluid domain consist of three distinct laws of conservation: the conservation of momentum, the conservation of energy and the conservation of mass. Note, that the following notation is influenced by the simulation framework FLUENT [1]. The momentum conservation law states that in a closed system the total momentum is constant. This applies to a non-accelerating reference frame. The law is defined as:

$$\left(\frac{\partial (\rho \vec{v})}{\partial t}\right) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\vec{\tau}) + \rho \vec{g} + \vec{F},$$

where $t$ is the time, $\rho$ the density, $\vec{v}$ the velocity vector, $p$ the static pressure, $\vec{\tau}$ the stress tensor, $\vec{g}$ the acceleration of gravity and $\vec{F}$ an external body force. The law of conservation of energy states that energy can neither be created nor destroyed but only be transformed. The law denotes:

$$\frac{\partial}{\partial t} (\rho E) + \nabla \cdot (\vec{v} (\rho E + p)) = \nabla \cdot (k \nabla T + (\vec{\tau} \cdot \vec{v})) + S_h,$$

where $E$ denotes the energy, $k$ the effective conductivity and $S_h$ the heat produced by the combustion.
The law of conservation of mass of a flow is determined as:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0.
\]  (6)

Additionally, the Reynold’s number \( Re \) is a key value in analyzing the fluid dynamics and in particular the heat transfer of a piston. It is defined as the ratio of inertial forces to viscous forces and is given by:

\[
Re = \frac{Qd}{\nu A},
\]  (7)

where \( Q \) denotes the volumetric flow rate, \( d \) the diameter, \( \nu \) the kinematic viscosity and \( A \) the inlet area. For small \( Re \), the flow becomes laminar and the Navier-Stokes equation \( \rho \dot{\vec{v}} = \rho \left( \frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla)\vec{v} \right) \) can be used. However, for greater \( Re \) turbulent flows are observed which may require different governing equations. These are Reynolds average Navier-Stokes equations or alternatively boundary layer equations.

The last main governing equation incorporates the definition of the Nusselt number \( Nu \). \( Nu \) is commonly utilized when looking at heat transfer problems since it represents the ratio of convective to conductive heat transfer normal to a boundary. \( Nu \) is defined by:

\[
Nu = \frac{hl}{k_f},
\]  (8)

where \( h \) is the convective heat transfer coefficient, \( l \) is the characteristic length of the boundary and \( k_f \) is the thermal conductivity of the fluid. In this particular problem, \( l \) is the length of the underside of the piston.

**Initial and boundary conditions**

Initial conditions of first or second kind can be used on the fluid and solid domain boundaries when using FEM and/or FVM for solving the equations above. Supplementary no-slip conditions can also be set for the fluid which guarantee that the fluid has zero velocity relative to the solid domain boundary.

Additionally, conditions on the interface between the solid and fluid domain are needed to ensure the continuity of the thermal fields. These are for the temperature field of the solid domain \( T^+ = T^- \) and for the heat fluxes of the flow \( q^+ = q^- \) where \((+)\) and \((-)\) denote points in the vicinity of the interface.

### 2.2 Movement of the piston

Two main approaches to simulating the movement of the piston are hereby applicable. Firstly, one might deform the mesh dynamically during the simulation. This approach proposes a morphing mesh which follows the actual movement of the piston and best resembles the physical setup. Therefore, a simulation with a
A conjugate heat transfer method for modeling oil-cooled pistons

Morphing mesh can provide the whole movement of the piston in one simulation process. However, morphing the mesh at every time step can be very computationally intensive.

On the other hand, there is the simulation of a stationary piston. The main idea is to avoid the morphing by simulating a single position of the piston only. Therefore, the mesh of the piston is fixed and can be used during the whole simulation process. This approach can, however, only model one state of the physical setup. Hence, simulations with a fixed mesh have to be reevaluated for various positions of the piston to resemble the actual movement of the piston.

3 Comparison of different methods

This section consists of a comparison between monolithic and partitioned approaches while a partitioned approach is chosen for this particular problem. From the various partitioned approaches available a conjugate heat transfer method is proposed for solving the heat transfer of an oil-cooled piston.

The monolithic approach can be categorized as a direct approach. The solution of a problem can therefore be obtained after solving the whole system matrix. As opposed to a monolithic approach, partitioned approaches are iterative methods which provide solutions after each iteration. These solutions might increase in accuracy with an evolving number of iterations. Nevertheless, enabling numerical convergence to the exact solution adds additional development effort for the setup of the model and can add additional computational costs if e.g. consistency checks are required between iterations. This demonstrates that the monolithic approach is by default numerically more stable than the partitioned approach.

Comparing the workload of both approaches, there are various advantages and disadvantages to both methods. The monolithic approach is costly in assembling the large system matrix. This is mostly due to the high memory capacity needed for storing all governing equations whereas the partitioned approach may demand memory for only one of the domains at any time during the simulation process. However, the boundary conditions have to be transferred after each solver execution which introduces additional communication efforts. Similar, the iterations need to be synchronized when using a loose coupling method. To overcome some of these disadvantages legacy solvers can be used. For the monolithic approach these are sparse matrix solvers like umfpack [3] or superLU [4] and for the partitioned approach frameworks like FLUENT [1] and KIVA [2].

In conclusion, the partitioned approach is chosen for its modularity and flexibility. Since there are only scalars needed to be transferred between both solvers the development of a transfer layer is simple compared to the assembling of a unified system matrix which is needed for the monolithic approach. Moreover,
it is very convenient to use sophisticated solid and fluid solvers which are well validated and optimized for performance. This optimization is also apparent when looking at the parallelization of the code. Depending on the size of the model, parallelization might be needed to solve realistic models. Additionally, the internal structure of neither solver interferes with the solution process for the partitioned approach resulting in a great choice of solvers.

The only scalar of interest for this work is the temperature field of the body of the piston. Since the proposed model omits the thermal deformation of the body, the modeling of the fluid forces and resulting deformations of the solid are to be excluded. Similar, when looking into stationary meshes the interface between the solid and fluid domain is constant. In summary, the only scalar which is needed to be transferred between both solvers is the temperature field. This leads to the choice for a conjugate heat transfer method which may be further improved in accuracy and stability with methods proposed e.g. in [13] and [8]. Although for this introductory work the transfer of the temperature field appears to be adequate.

4 Conjugate heat transfer method

This section elaborates on the conjugate heat transfer method presented for modeling oil-cooled pistons and is divided into two main parts. The first part describes the simulation process while the second part elaborates on the interaction between solvers.

The proposed simulation procedure is depicted in figure 2. First, the piston and nozzle geometry is crafted with the help of computer-aided designing methods. A mesh of the moving piston can then be derived from these geometries. From this step onward the solid and fluid domain are separated except of the transfer of boundary conditions. For the solid domain a finite element method can be used to simulate the thermal conductivity of the body of the piston. For the fluid domain a computational fluid dynamics method can be applied to simulate the temperature and flow field. This process is highly-influenced by [5].

Figure 3 shows the interaction of both solvers in detail [12]. In this example, the fluid solver is executed first. Since the solid solver has not been executed yet, the wall temperatures of the piston are unknown. The wall temperatures denote the temperatures at the common boundary of both domains and the lower insulated boundary. These unknown boundary conditions are initially estimated. The fluid solver then computes the heat fluxes over the common boundary which are transferred to the solid solver. The solid solver uses the heat fluxes as boundary conditions and computes the wall temperatures. The wall temperatures are thereafter transferred to the fluid solver which completes the first iteration. Fur-
A conjugate heat transfer method for modeling oil-cooled pistons

**Fig. 2.** The proposed simulation process

Furthermore, one could check for consistency after each solver execution or every iteration (see chapter 3).

**Fig. 3.** The proposed conjugate heat transfer method
5 Possible challenges

When implementing the proposed conjugate heat transfer method one could face several challenges. The intention of this section is to discuss some of the challenges and propose methods on how to overcome them. The main difficulties come from:

- boundary conditions
- synchronizing the fluid and solid solvers
- parallelizing of the CHT method
- meshing the domains.

5.1 Initial estimate of boundary conditions

A challenge which is often observed in partitioned conjugate heat transfer methods is the importance of the initial estimation of boundary conditions [10]. The rate of convergence of the method highly depends on this estimation. There are several methods specialized on finding initial boundary conditions. These include experimenting and typical values observed by engine manufacturers. These methods are highly problem-specific, though, and do not provide general applicability. The fall-back method in case the approaches above fail is trial and error.

5.2 Synchronization strategies

The fluid and solid solver and the corresponding transfer of boundary conditions is usually synchronized for a better control over the solving algorithm. In addition to that, synchronization is used to better predict key characteristics of the solvers like rate of convergence, termination, etc.

Let $\alpha_s \tau_s$ with a characteristic time $\tau_s$ for the heat propagation through the solid be the quantity of which the solid is advanced in time in one iteration. Similar for the fluid domain, let $\alpha_f \tau_f$ with a flow characteristic time $\tau_f$ be the quantity of which the fluid is advanced in time in one iteration.

There are three main issues to focus on synchronization [12]:

- Synchronization in physical time: To synchronize both solvers in physical time the solid and fluid have to advance of the same quantity every iteration. Therefore $\alpha_s \tau_s = \alpha_f \tau_f$ has to hold.
- Synchronization of the convergence rate: To guarantee that both solvers reach steady state at the same time both factors $\alpha_s$ and $\alpha_f$ are set to the same value. Note, that when $\alpha_s = \alpha_f$ holds both solvers are in general not being executed at same time steps.
- Synchronization in CPU time: On parallel machines two main strategies can be used which will be discussed in the next section. To parallelize the execution of the solid and fluid solver a synchronization in CPU time is chosen and required that both solvers are being executed in the same wall-clock...
time. In this context this time can be the real time elapsed in one iteration of the algorithm on a parallel machine or the CPU time consisting of the accumulated time for the execution of one iteration of each of the processing units. Note, that $\alpha_s$ and $\alpha_f$ have to be adjusted after each iteration to enforce same timings for both solvers at every iteration.

5.3 Parallelization strategies

When looking at realistic problems, parallelization might be required to handle the size of the problem. There are two main parallelization strategies [12] discussed in this section:

- Sequential Coupling Strategy (SCS): If the used legacy solver can be executed in parallel a sequential coupling strategy can be applied. This way, the solvers are loaded on the parallel machine sequentially while being executed in parallel on all available processing units $N$.

- Parallel Coupling Strategy (PCS): Since the proposed conjugate heat transfer method uses a partitioned approach both solvers are coupled as loosely as feasible. This means that the exchange of a single scalar is the only transfer required between both solvers. Hence, both solvers can be executed in parallel while keeping the exchange of boundary conditions as proposed above. The only adjustment needed is the estimation of the initial boundary conditions of both solvers. These are the wall temperatures $T^{(-1)}$ and the heat fluxes $\Phi^{(-1)}$. In this case, both solvers share all available processing units $N$.

To achieve good load balancing the proposed synchronization in CPU time may be used to distribute the processing units for each solver.

5.4 Mesh generation

The solid and fluid domain can be meshed separately in a partitioned approach. This can be advantageous since the fluid problem usually requires finer meshes compared to the solid mesh to achieve similar numerical convergence. The boundary nodes of both domains may then be non-matching in terms of their spatial position. Thus, the boundary values of one domain need to be interpolated to fit to the corresponding nodes of the other domain. The downside of this interpolation is that it introduces additional numerical error to the computation.

Matching grids may instead be employed to overcome this additional inaccuracy. The meshing of both domains is then initialized with matching boundary nodes. From there, a basic grid is computed on both domains according to each of the methods applied to both domains. Refinement strategies might be incorporated to further improve each grid. The drawback of matching grids is that both domains need to be meshed at the same time. Changes to one of the domains result in the remeshing of both domains. This is especially computationally intensive if the body of the piston changes its size over time due to thermal expansion which is however assumed to be negligible in this work.
6 Conclusion

In conclusion, the problem was identified as a fluid-structure interaction problem and a corresponding model of the oil-cooled piston was derived. Different methods for solving FSI problems were compared and the conjugate heat transfer method was chosen for its modularity and simplicity. Finally, some of the typical challenges with CHT methods were evaluated in regards to this particular problem. In a next step, the proposed method may be implemented and validated with analytical solutions and experimental setups. Additionally, further investigations on the best suitable method of FSM may be carried out. After the model is validated it might be put into use by engine manufactures to improve the overall engine development cycle.

References

A conjugate heat transfer method for modeling oil-cooled pistons