Fast design evaluations of automotive Oil Coolers considering a porosity approach

Dr. Carlos Massaiti Okubo Júnior
Sergio Adrian Ferreyra
MAHLE

e-mail: carlos.c.junior@mahle.com; sergio.ferreyra@mahle.com

ABSTRACT

The advances in computational simulation tools bring the possibility of analyzing projects and products in a very detailed manner. Techniques like Computational Fluid Dynamics allow to thoroughly evaluate the flow inside components of interest. In this context, it is important to find balance between how detailed an analysis must be and its time to deliver. Components like the automotive oil coolers have complex geometries, a big volume of details and dimensional features that may have orders of magnitude of difference. All these together make its simulation a very time demanding process. In this work, an approach is presented with the objective of reducing the time taken to calculate the pressure drop by CFD. The proposed strategy is based on the porous media modelling. A study case is shown, highlighting the results obtained by the porous media model against the results obtained from a detailed analysis that considers all the geometric characteristics of the component. Focus is given on the accuracy of the results and also on the time taken for each analysis.

INTRODUCTION

Automotive oil coolers are compact heat exchangers used in automotive applications (see Figure 1). The main function of these components is to remove the heat from the oil, commonly used in Internal Combustion Engines (ICE) and transmissions, ensuring longevity and reliability of the mechanical components.

Oil coolers are usually composed of stacked metallic plates. Between each pair of plates, there can be complex finned geometries (fins are usually called turbulators, see Figure 2) and compact narrow flow passages. The presence of turbulators aims to maximize the surface area and promote a turbulent flow, both enhancing the heat transfer. With more surface area, often comes higher pressure losses inside the components, so the balance between heat transfer and pressure loss inside an oil cooler is a frequent challenge in the project of these parts.

Figure 1. Oil Cooler for automotive application [1].

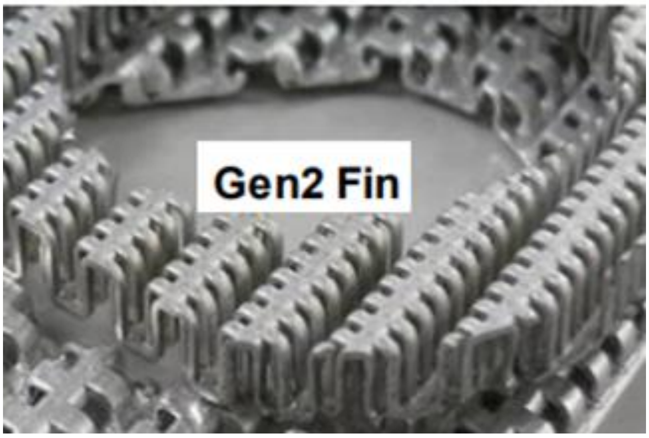

Figure 2. Oil Cooler turbulators.

The assessment of heat transfer and pressure drop characteristics of an oil cooler in a design stage is often performed with the use of Computational Fluid Dynamics (CFD) techniques. The more complex the component is, the more challenging the CFD calculations become and also the more time it consumes. Depending on the complexity of the geometry and on the computational resources available, the simulations can take from hours to days [3].

In the case of oil cooler design, it is not uncommon to go over a couple of design iterations until the balance between heat transfer and pressure drop is acceptable for a given project. Hence, the time used in each simulation can become a significant advantage in terms of cost savings and results delivery when designing an oil cooler.

In this context, simplification techniques can be very advantageous. When geometries have small features and complicated patterns, as in the case of oil coolers, the use porous media modeling arises as an interesting option. The region with the complicated geometry is removed from the model and replaced by a part with no internal details. This part is then defined as a porous region in the model and, by using the correct porosity coefficients, the complexity of the calculations is drastically reduced while keeping an acceptable accuracy. In Figure 3, an example of a model with all the details and a model with the porous media simplification are presented.

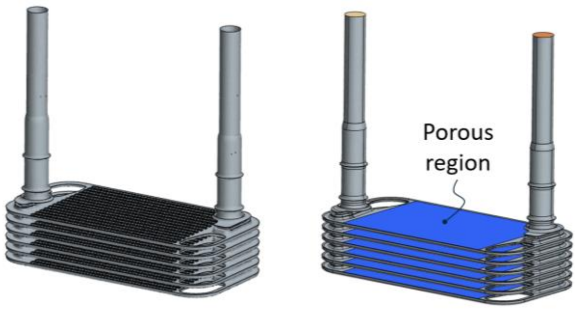

Figure 3. Detailed model (left) x porous model (right).

In this work, a comparison between a detailed CFD calculation with the results from a simplified CFD model is presented. The objective is to calculate the pressure drop of the geometry for given flow conditions (heat transfer is not evaluated in this work). In the following sections, the methodology is described, explaining briefly the fundamentals of porous media modelling in CFD. Then, the case study is explained with details on geometry, mesh, boundary conditions and both analyses, the detailed calculation and the simplified one with the porosity region. The comparison between the two calculations demonstrates that the time savings together with the acceptable accuracy of the simplified model can be used to reduce the time in the development in a safe manner.

METHODOLOGY

In this section, the methodologies used for the pressure drop evaluations are described. The evaluation of the detailed geometry is performed in a "traditional" manner, where the geometry with all its details is imported in the software and the flow inside this geometry is calculated. For the flow in porous media, an additional step needs to be performed, which is the calculation of the porous coefficients. These depend on the geometry of the turbulators and on the fluid properties, so they must be calculated for each case.

CFD FOR PRESSURE DROP EVALUATIONS

In the detailed analysis for pressure drop, the flow is considered steady state, with constant density and energy equation is not solved, so the Navier-Stokes equations, Equations (1) and (2), are considered in the following form [2, 3]:

$$\nabla \cdot \boldsymbol{u} = 0 \tag{1}$$

$$\rho \nabla \cdot (\boldsymbol{u}\boldsymbol{u}) - \nabla \cdot (\mu(\nabla \boldsymbol{u} + \nabla \boldsymbol{u}^T)) + \nabla p = 0 \tag{2}$$

Where ρ is the fluid density, $\boldsymbol{u}$ is the local velocity, μ stands for the viscosity and p represents the local pressure.

The whole procedure involves the geometry import, the mesh creation, the application of boundary conditions and fluid properties, the solution of the flow problem and the post processing. In Figure 4, a flowchart with all the steps is presented.

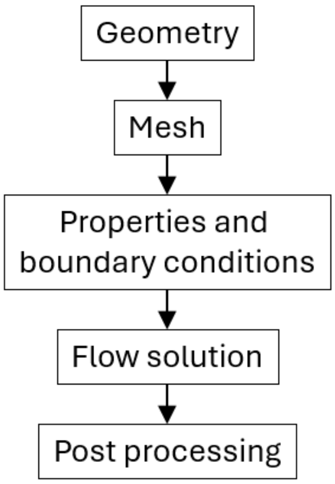

Figure 4 - Detailed CFD flowchart.

POROUS MEDIA REPRESENTATION IN CFD

The porous media approach in CFD introduces additional resistance terms to the momentum equations, allowing the model to capture the macroscopic effects of internal obstructions such as fins, turbulators, and narrow channels. The equations modified to include porous media resistance take the following form [4]:

$$\nabla \cdot \boldsymbol{u} = 0 \tag{3}$$

$$\rho \nabla \cdot (\boldsymbol{u}\boldsymbol{u}) - \nabla \cdot (\mu(\nabla \boldsymbol{u} + \nabla \boldsymbol{u}^T)) + \nabla p - f_p = 0 \tag{4}$$

The last term, f_p, is the local resistance added by the porosity term and has two components [4]:

$$f_p = P_v \cdot \boldsymbol{u} + |\boldsymbol{u}| P_i \cdot \boldsymbol{u} \quad (5)$$

Two types of losses are considered. The tensor P_i accounts for the inertial losses and is proportional to the square of the velocity. The term P_v is responsible for the viscous losses and is linear with the local velocity [4]. Both tensors, P_v and P_i, are composed by coefficients that are obtained from models of portion of the turbulator region.

Porous media can be modeled as isotropic or anisotropic. For Oil coolers simulation, anisotropic porous media is used, allowing accurate representations of the flow resistance in different directions.

In the porous media cases, the sequence of steps changes when compared to the previous one, because it is necessary to calculate the inertial and viscous coefficients in different directions. Thus, the first part of the approach using porous media is to extract a small portion of the turbulator region, calculate the porous coefficients (with the "traditional" CFD approach shown in Figure 4), then import the geometry (without all the detailed information on the turbulators), assign the porous representation and follow the steps described in the previous section for the pressure drop evaluation. In Figure 5, the flowchart for the porous media modelling is presented.

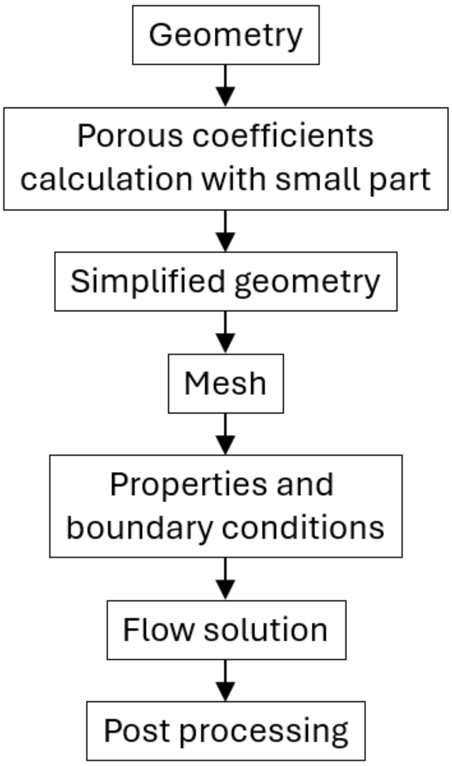

Figure 5 - Flowchart with porous media.

CASE STUDY

The case study for this work is the coolant side of an automotive oil cooler. Six operating conditions assessed: two sets of fluid properties (Case A and Case B) with three different mass flows for each one. The detailed model is meshed and calculated with all its features and complexity. In the simplified model, the turbulators region is substituted by a solid where the porosity coefficients are applied. The detailed geometry and the simplified one are the same already seen in Figure 3. All calculations are performed in a Linux machine of 16 cores and 384 GB RAM, with the commercial software StarCCM+, version 2022.1 (17.02.007-R8).

GEOMETRY

The inlet and outlet sides and the pressure measurement sections are presented in Figure 6.

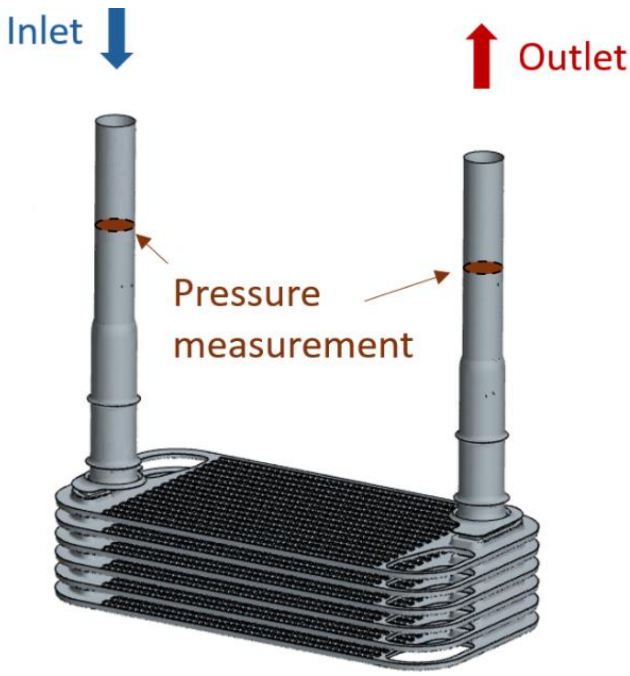

Figure 6. Oil cooler inlet, outlet and measurement planes.

MESH

In both cases, the detailed model and simplified one, the mesh used is the polyhedral type with a base size of 1 mm. The final cell count for the detailed model is approximately 37 million cells, while, for the simplified model, it is approximately 9.2 million cells. In Figure 7 and Figure 8, some sections are presented.

Figure 7. Mesh (detailed model).

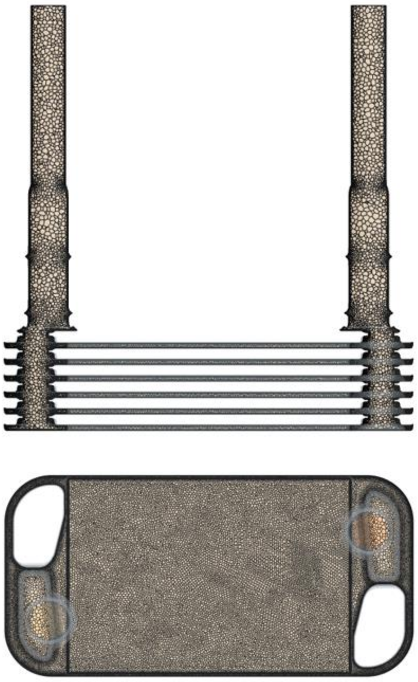

Figure 8. Mesh (porous model).

PROPERTIES AND BOUNDARY CONDITIONS

The fluid properties and boundary conditions used are presented in Table 1. The densities, viscosities and mass flow values are based in real operating conditions for the oil coolers.

Table 1 - Fluid properties and BCs for cases A and B.

Property	Case A	Case B
Density (kg/m³)	ρ	0,85 x ρ
Dyn. visc. (Pa.s)	μ	0.42 x μ
Massflow 1 (kg/s)	M_A	M_B
Massflow 2 (kg/s)	4.00 x M_A	4.00 x M_B
Massflow 3 (kg/s)	10.00 x M_A	10.00 x M_B

DETAILED CFD ANALYSIS

The detailed CFD case ran for 1500 iterations (for each operating condition). Convergence was considered when the pressure drop values became stable with respect to the iterations. Each mass flow calculation time took approximately 8.4 hours of calculation.

In Table 2, the pressure drop values (ΔP) for each case are exhibited. In Figure 9 and Figure 10, the static pressure contour and the streamlines are presented for the highest massflow of Case A.

Table 2 - Pressure drop values for detailed CFD.

	Case A	Case B
ΔP for massflow 1 (kPa)	0.55	0.34
ΔP for massflow 2 (kPa)	5.15	3.79
ΔP for massflow 3 (kPa)	27.19	21.60

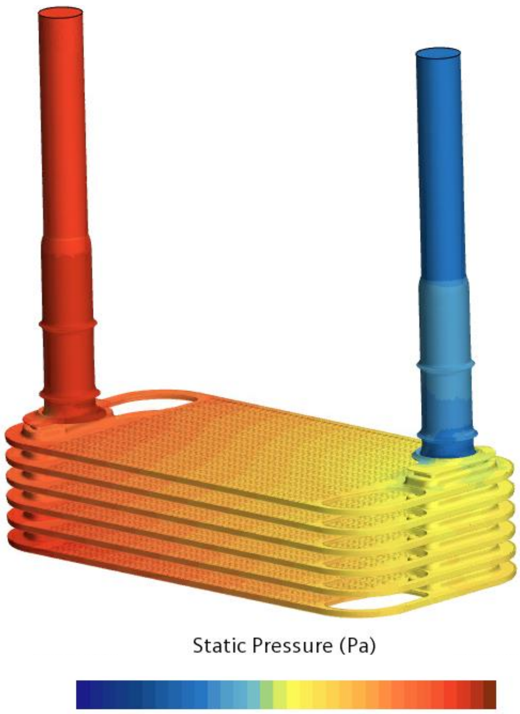

Figure 9. Static Pressure (detailed model).

Figure 10. Streamlines (detailed model).

POROUS COEFFICIENT CALCULATIONS

For the simplified model, before the pressure drop calculations, it is necessary to calculate the porosity coefficients. A small part of the turbulator region is extracted (see Figure 11). Different mass flow values are passed through the small part in the main direction and also in the cross direction (perpendicular to the main one) and the pressure drop for each case is calculated. All the pressure drop values are arranged in a graph of linear pressure drop (y axis) in function of mean velocity (x axis). Then, a second order curve is fit into the data and the inertial and viscous coefficientes can be extracted. In Figure 12, the graph for the main direction with fluid properties of Case A is presented.

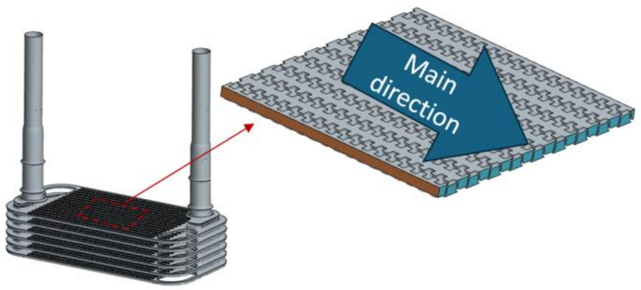

Figure 11. Turbulators part (main flow direction).

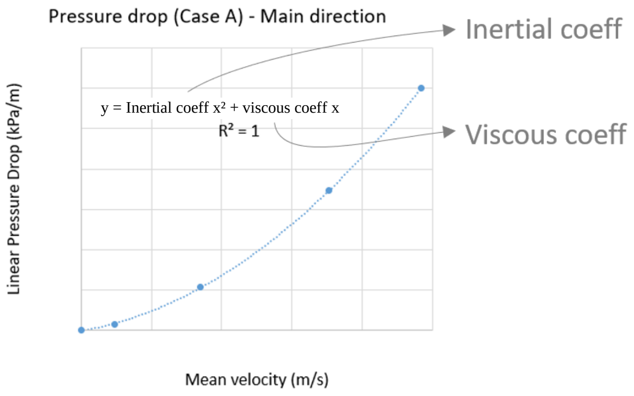

Figure 12. Porous coefficients for Case A in main direction.

In each direction and for each fluid property, 4 mass flows are used, so the number of CFD calculations to get all the necessary porosity coefficients is 16 (2 directions with 2 different fluid properties). All the coefficients are presented in Table 3.

Table 3. Porous coefficients.

Fluid properties	Direction	Inertial coeff	Viscous coeff
Case A	Main	C_I	C_V
Case A	Cross	13.6 x C_I	2.5 x C_V
Case B	Main	0.9 x C_I	0.4 x C_V
Case B	Cross	11.6 x C_I	1.7 x C_V

SIMPLIFIED MODEL

Having the porosity coefficients, the turbulators region is replaced by a solid and represents a porous part of the model. The coefficients are inserted considering the appropriate coordinate system to perform the calculations. For each operating point, 1500 iterations are used. Each case took approximately 1.9 hours. The results are presented in Table 4. Static pressure contours and streamlines can be seen in Figure 13 and Figure 14.

Table 4 - Pressure drop values for porous media model.

	Case A	Case B
ΔP for massflow 1 (kPa)	0.60	0.37
ΔP for massflow 2 (kPa)	5.32	3.91
ΔP for massflow 3 (kPa)	27.36	21.75

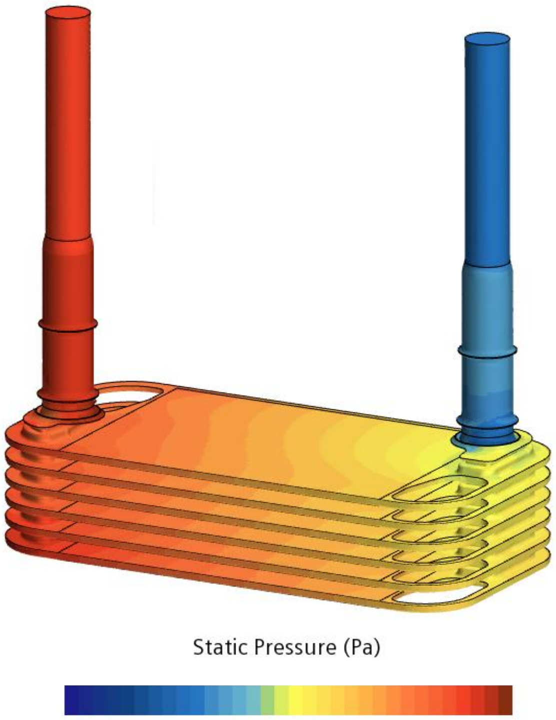

Figure 13. Static pressure (simplified model).

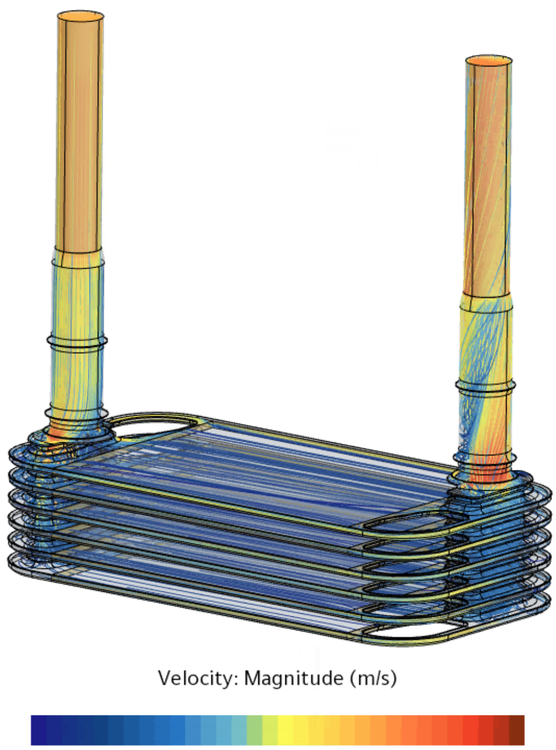

Figure 14. Streamlines (simplified model).

COMPARISON

The pressure drop values for the detailed model (Table 2) and for the simplified model (Table 4) are compared and the error with respect to the detailed model is presented in Table 5. The time comparison for the models is presented in Table 6.

As can be seen, the accuracy for the presented cases is below 9% and is improved as the mass flow increases, reaching values below 1%. The total time for each approach can be seen in the last two rows of Table 6. For this example, the time used in mesh generation and flow solution is reduced by approximately 76% when using the porosity approach.

Table 5 - Error in % with respect to the detailed model.

	Case A	Case B
Error for massflow 1	8.9%	7.5%
Error for massflow 2	3.3%	3.2%
Error for massflow 3	0.6%	0.7%

Table 6 - Times for each case.

Model	Activity	Time (hours)
Detailed	Mesh generation	0.70
Detailed	Flow calculation	8.4 (6x)
Porous coeffs	Mesh generation	0.033 (2x)
Porous coeffs	Flow calculation	0.038 (16x)
Simplified	Mesh generation	0.12
Simplified	Flow calculation	1.9 (6x)
Detailed	Total time	51.1
Simplified	Total time	12.2

CONCLUSIONS

This work highlights the value of porous media modeling in CFD simulations of oil coolers, particularly in contexts where computational resources or time constraints are limiting factors. By representing a geometrically complex component as a porous region, simulations can be performed with reduced computational effort when compared to the resources required by fully resolved geometries. In the cases presented, the accuracies of the simplified models (with respect to the detailed model) varied from approximately 9% to less than 1%, while the reduction in computational time obtained is about 76% for the six points calculated.

The comparison between the detailed approach and the porous media approximation exhibits a clear trade-off: while the detailed model provides greater insight into local flow phenomena, the porous model delivers results significantly faster with acceptable accuracy for design evaluations in intermediate phases. This balance makes the porous media modeling an interesting choice for early stage development and design optimization loops as the reduction in computational demand facilitates parametric studies and rapid iterations, supporting a more agile development process.

In conclusion, porous media modeling appears as a strategic simplification technique that brings efficiency, flexibility, and agility to the CFD analysis of Oil coolers, facilitating and accelerating a design development.

Future work may include refining porous coefficients calibration strategies, integrating conjugate heat transfer and developing the methodology to more advanced simulations, where the oil and coolant are present in the same model.

REFERENCES

1. THERMAL MANAGEMENT. Access in: April 24th 2025. Available in: <https://www.mahle-aftermarket.com/na/en/products-and-services/engine-cooling-&-air-conditioning/>

2. MOUKALLED, F.; MANGANI, L; DARWISH, M. *The Finite Volume Method in Computational Fluid Dynamics: And Advanced Introduction with OpenFOAM*, 2016.

3. FERZIGER, J. H.; PERIC, M.; STREET, R. L. *Computational Methods for Fluid Dynamics*, 4th edition, 2020.

4. Siemens Digital Industries Softwre, *Star-CCM+ User Guide*, 2021.