24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-2120
DATA TRANSITION FROM 3D TO 1D SIMULATION FOR UNDERHOOD ANALYSIS OF COOLING SYSTEM

Ricardo de Oliveira Silva
Rogério Gonçalves dos Santos
Universidade de Estadual de Campinas – Unicamp, Faculdade de Engenharia Mecânica – FEM, Campinas, SP, Brazil
r065424@dac.unicamp.br
roger7@fem.unicamp.bre-mails

Abstract. *The employment of 1D simulation via block diagram and 3D simulation using CFD (Computational Fluid Dynamics) is usual in the main development centers of at automotive industry in the world, when is applied on underhood analysis, for cooling systems sizing. The present work aims to show a 1D model and its data extracted from a 3D model which was simulated via CFD and deals on the transition of these data. Although CFD simulation has sophisticated resources of calculation, in terms of fluid mechanics and heat transfer, it is known that this kind of simulation requires substantial time to prepare models and resources of data processing; such peculiarity is not observed on 1D simulation. However, 1D method cannot get simulates the aerodynamics effects of air flux from the ambient passing through opened grilles, the underhood until the outlet of the engine compartment. Therefore, 1D calculations method needs a complementary help of CFD in terms of fluid mechanics results, reducing the time of processing and leaving the thermal analysis for 1D calculation. The results obtained from this kind of approach have shown rather accuracy with experimental data what reinforces the best practices of engineering in terms of low cost and time of development.*

Keywords: *Simulation, cooling system, discretization, heat exchanger, thermal analysis*.

1. INTRODUCTION

The conversion of an air flux passing through the engine compartment is not an easy task to of a vehicle in a 1D representation.

The Fig. 1 shows different approaches of underhood analysis. For 1D analysis, the air flow distribution takes on non-predictive characteristics, and because of that this approach does not consider backpressures, leakages and recirculation, thus it is usual that 1D software of calculation has a method of flow discretization, with support of a graphical interface, which places the elements of the cooling system into a flow space, transforming this space in a discretized matrix, similar what occurs in a CFD analysis, where the dominium or zone are analyzed in a 3D mesh, which is formed by cells or control volumes, resulting on an accurate idea of air flow distribution.

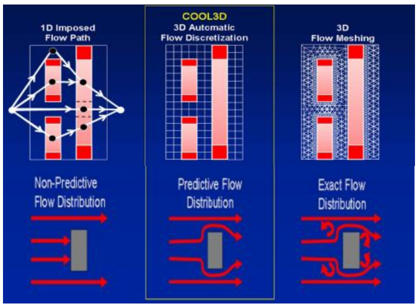

Figure 1. Different approaches of air flux.

Different from what is proposed by CFD, the predictive flow distribution does not analyze the recirculation and turbulence, becoming this kind of approach totally dependent of additional information of CFD's approach.

The Fig. 2 (a) shows an example of flow space and its elements of cooling system which were modeled and created from a graphical interface of an 1D software. This representation is converted on a 3D matrix, where each element undergoes a process of discretization, such as it is shown in the Fig. 2(b).

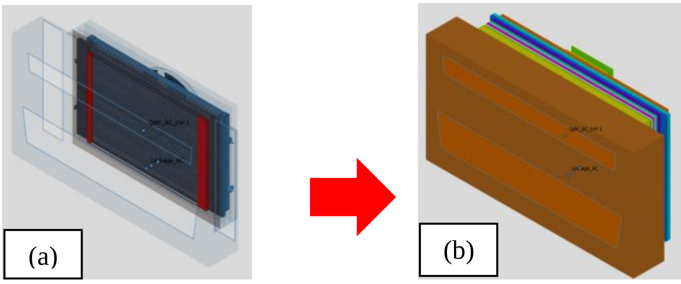

Figure 2. An underhood model (a) is converted in a discretized model (b) for 1D analysis.

2. TRANSITION BETWEEN 3D GEOMETRY DATA TO A 1D MODEL

The present work recommends that the first step to create an underhood model, for cooling system analysis, should be performed from of flow space modeling, which should follow the main dimensions of the vehicle.

This sizing can be obtained directly from CAD or via CFD model, just like it is shown in the Fig. 3. This is the first interaction between the models 3D and 1D.

The proposed 1D model requires an analysis which starts from the frontal plane of the bumper, near the grilles, until the fan plan plus 20 mm distance.

It is noticed that, when a CFD analysis is used to support 1D simulation, a complex geometry, like an engine for example, can be dispensed of a conversion to 1D, reducing the size of the 1D model, as it will be shown during the present work.

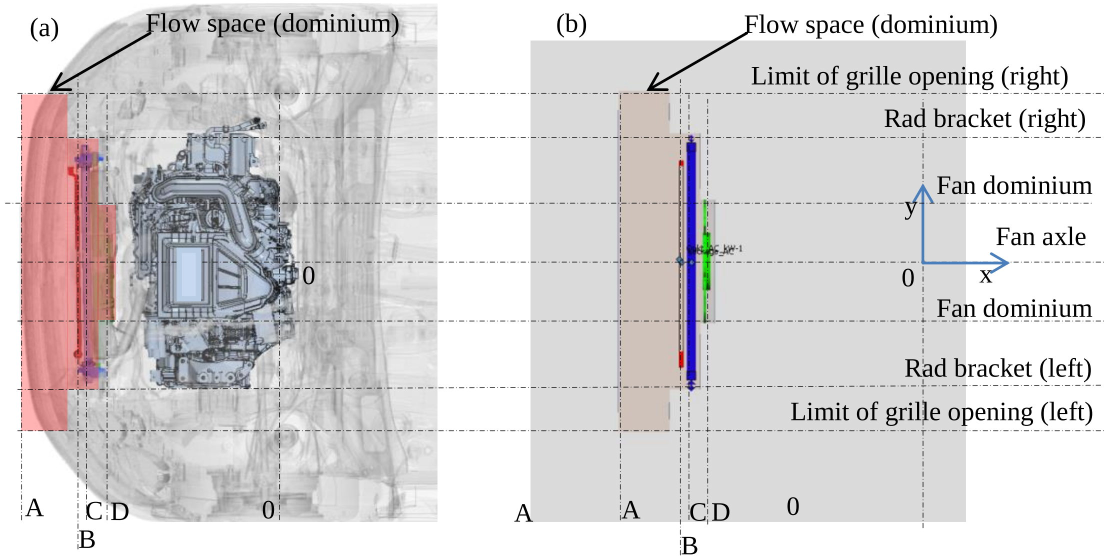

Figure 3. Top view of the flow space: (a) 3D model – CAD or CFD model; (b) 1D model – Graphical interface

In the Fig. 3 and 4, A, B, C and D represent:

- A- Frontal plan;
- B- Condenser plan;
- C- Radiators plan (main and auxiliary for intercooler system);
- D- Fan plan;

The Fig. 4(a) shows a right side view of a section plan passing through y=0m a correlate view of 1D model is shown in the Fig. 4(b).

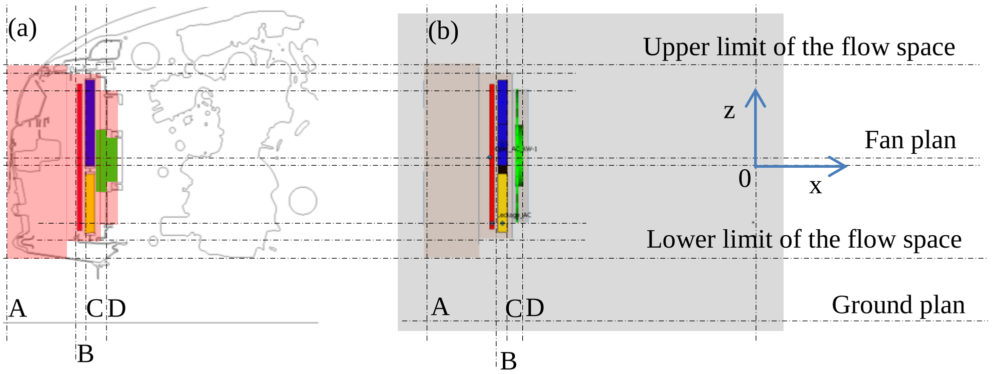

Figure 4. Right side view of the flow space: (a) 3D model – CAD or CFD model; (b) 1D model – Graphical interface

Openings (inlets/outlets) represent other important simplification of an underhood analysis, as it is shown in the Fig.5 (a), (b), (c) and (d).

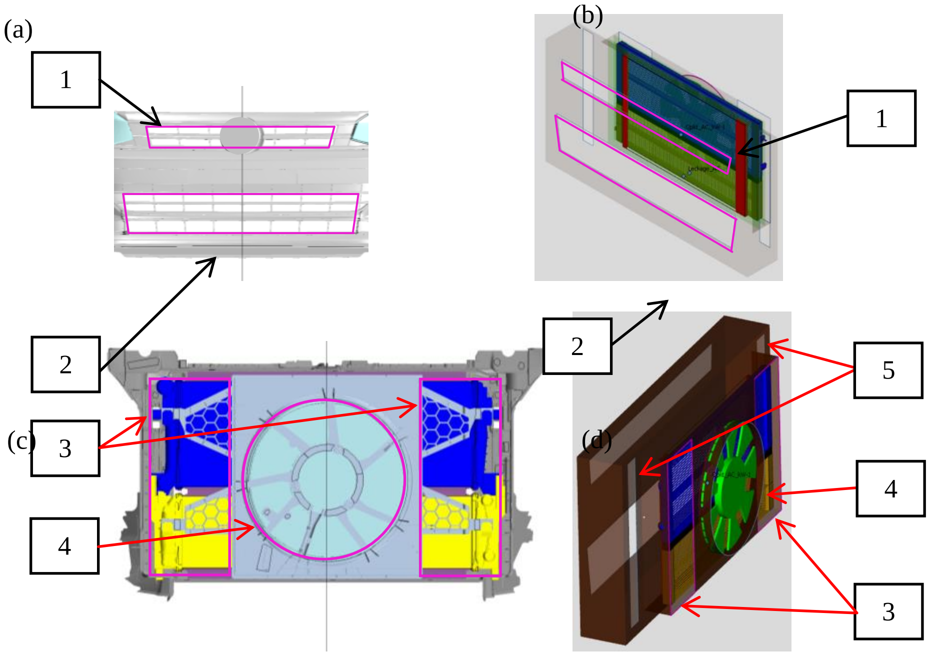

Figure 5. Openings on the flow space (a) Frontal grilles, (b) Frontal openings of the Flow Space, (c) Shroud set and (d) Rear openings of the Flow Space. (1) Lower grille opening; (2) Upper grille opening; (3) Shroud's openings; (4) Fan opening; (5) Leakage openings.

The external profile of each opening is enough to characterize the items: 1, 2, 3 and 4, as it is shown in Fig 5. For the item 5 (leakage) is necessary to analyze the difference between the total air mass flow rate which passes through ($\dot{m}_{rad}$) and around ($\dot{m}_{a\_rad}$) the radiators and what is passing through the grilles ($\dot{m}_{grilles}$). These values should be obtained from CFD, where virtual sensors are created and positioned on each component analyzed, in complement, it is rather important to inform, that the present work was made in steady state and constant air density (ρ) using k-epsilon (k-ε) as turbulence model.

Where:

$$\dot{m}_{leakage} = \dot{m}_{grilles} - \dot{m}_{rad} - \dot{m}_{a\_rad} \tag{1}$$

And:

$$\dot{m}_{leakage} = \rho.A_{leakage}.v_{leakage} \quad (2)$$

Wherefore:

$$A_{leakage} = \frac{\dot{m}_{leakage}}{\rho.v_{leakage}} \quad (3)$$

The air velocity in the leakage ($v_{leakage}$) from which is considered the same of the vehicle and the air passage occurs through two rectangular openings, being that its height ($h_{leakage}$) is obtained directly from the flow space, thus it is obtained from the length (L_{eakage}).

$$A_{leakage} = L_{leakage}.h_{leakage} \quad (4)$$

The heat exchangers and others components are positioned into the modeled flow space with information extracted from de CAD model.

The Fig. 6 (a) shows two heat exchangers a main radiator (in blue) and an intercooler (in yellow). Additionally, there is in this picture a gasket (in black) between the main radiator and the intercooler. The Fig 6 (b) shows the assembly of the heat exchanger including a condenser (in red).

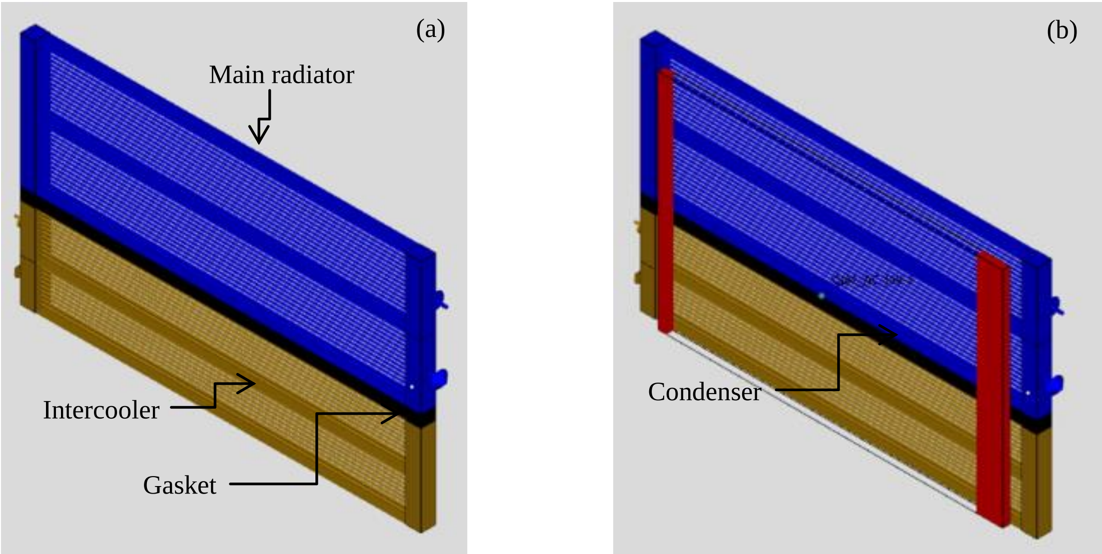

Figure 6. Examples of heat exchangers used in an underhood model: (a) Radiators and a (b) Condenser

It is possible in this phase of development to setup the characteristics of each heat exchanger, their geometry, which includes: i) length, ii) height, iii) width, iv) diameter of mouths, v) number of tubes, vi) fin pitch, the format of fins, tubes and tanks, and physical properties like i) curves of performance, ii) backpressure and iii) materials for the heat exchangers, the initial condition of the coolant side and the air side. Usually, the characteristic curves of each heat exchanger are supplied by each respective supplier via experimental tests or via calculation

Similar procedures are adopted for fan modeling which includes diameter of fan, number of blades, hub diameter, depth and the characteristics curves of the fan. In other words, the air mass flow rate passes through the fan in function of the pressure drop. The Fig. 7 shows a fan modeled for 1D discretization.

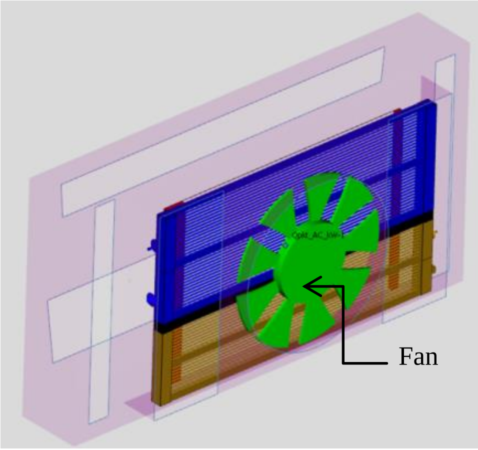

Figure 7. Fan representation

The air flow distribution undergoes direct influence of the layout of underhood components, but to become this representation possible, it is necessary to add the effects of backpressure of air around the heat exchangers.

This effect is obtained creating a backpressure plan around the heat exchangers. These plans are located in the middle of the heat exchangers, as it can be seen in the Fig. 8(a).

A similar procedure is performed on CFD, but in this case a virtual sensor is created in order to obtain the average of total pressure (P_t) (eq. 5) of this dominium, an example of this approach can be seen in the Fig. 8(b).

$$P_t = P_s + P_d \qquad (5)$$

Where: P_s is the average of static pressure and P_d is the average of dynamic pressure around the heat exchangers, both variables are applied on the surface in analysis.

During the process of calculation, the pressure around the heat exchangers is monitored, thus if it is necessary, the model 1D is corrected, in order to generate the same physical effect calculated of CFD simulation.

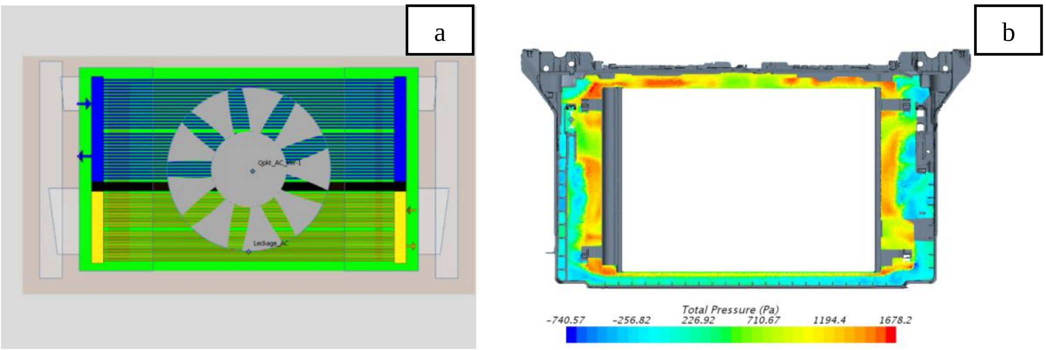

Figure 8. Plans around the heat exchangers.
(a) Backpressure plan; (b) CFD's scene of total pressure field around the condenser.

3. DISCRETIZATION AND CONVERSION FOR 1D MODEL

So far, each one element contained in this underhood analysis was positioned and their main characteristics were input in the software, using a 3D interface of visualization, this procedure has allowed the definition of the air side, which is contained in this simulation, where each element is converted in a matrix 1D, as it is shown in the Fig. 9 (a).

Additionally, the Fig. 9 (b) shows the matrix, where the discretized elements were converted in numbers (nodes, elements, connections, etc.).

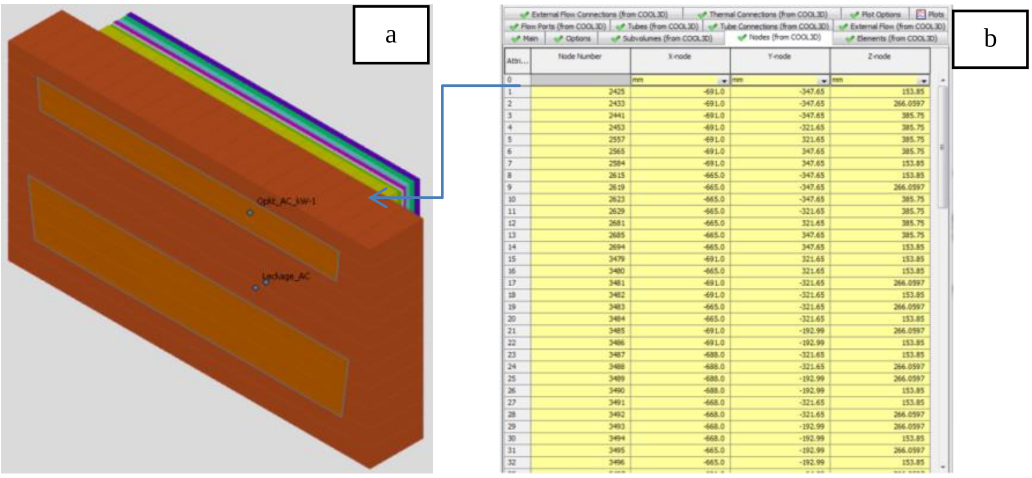

Figure 9. (a) Graphical representation of an underhood analysis by discretization/
(b) Matrix representation of a heat exchanger.

In comparison of a CFD dominium which contains $12x10^6$ cells (Fig. 10), the present discretized model has just 2434 cells Fig. 9(a), therefore the computational time of processing is reduced substantially, and it is not requested an equipment of higher capacity of processing in comparison with a CFD analysis. Another point which should be highlighted; is the CFD analysis including a complete vehicle, for 1D analysis the model is reduced. Needless to say, if a thermal analysis was performed via CFD, the capacity of processing would be rather complex, due to highest number of new variable and care with the model.

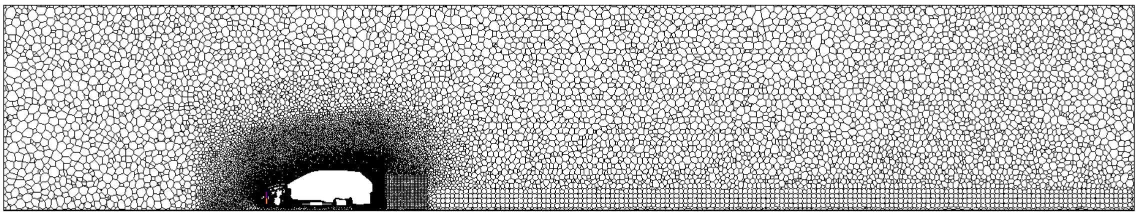

Figure 10. CFD Dominium for a complete vehicle

The Tab. 1 shows an overview of one input data for the proposal of underhood analysis.

Although the focus of the present work is not to teach how to input data, it is recommended higher care, when these data are inserted in the software.

Table 1. Analyzed elements in a predictive air flow distribution

Component	*Main Input Data*	*Source*
Upper Grille inlet	Vehicle speed	Avaliable data
	Ambient condition	(ambient temperature, atmospheric pressure and humidity)
Lower Grille inlet	Vehicle speed	Avaliable data
	Ambient condition	(ambient temperature, atmospheric pressure and humidity)
Leakage (left)	Resources of software 1D	Best point of operation
Leakage (right)	Resources of software 1D	Best point of operation
Condenser	Rejected heat constant	Mesured data
	Condenser curve of drop pressure (air side)	Mesured data
Back Pressure plan for Condenser	Resources of software 1D	Best point of operation
Intercooler Radiator	Characteristcs curves of fan	Mesured data
Back Pressure plan for Intercooler Radiator	Resources of software 1D	Best point of operation
Radiator	Characteristcs curves of fan	Mesured data
Back Pressure plan for Radiator	Resources of software 1D	CFD Data
Shroud oppenings	Outlet pressure	CFD Data
Fan	Characteristcs curves of fan	Measured (differential of Pressure x air flow rate)
Fan outlet	Outlet pressure	CFD Data

The Fig. 11 shows the result of a conversion of an underhood model which was worked via a graphical interface in a 1D block diagram. In dashed red line are the interfaces of this model with the ambient and its physical properties.

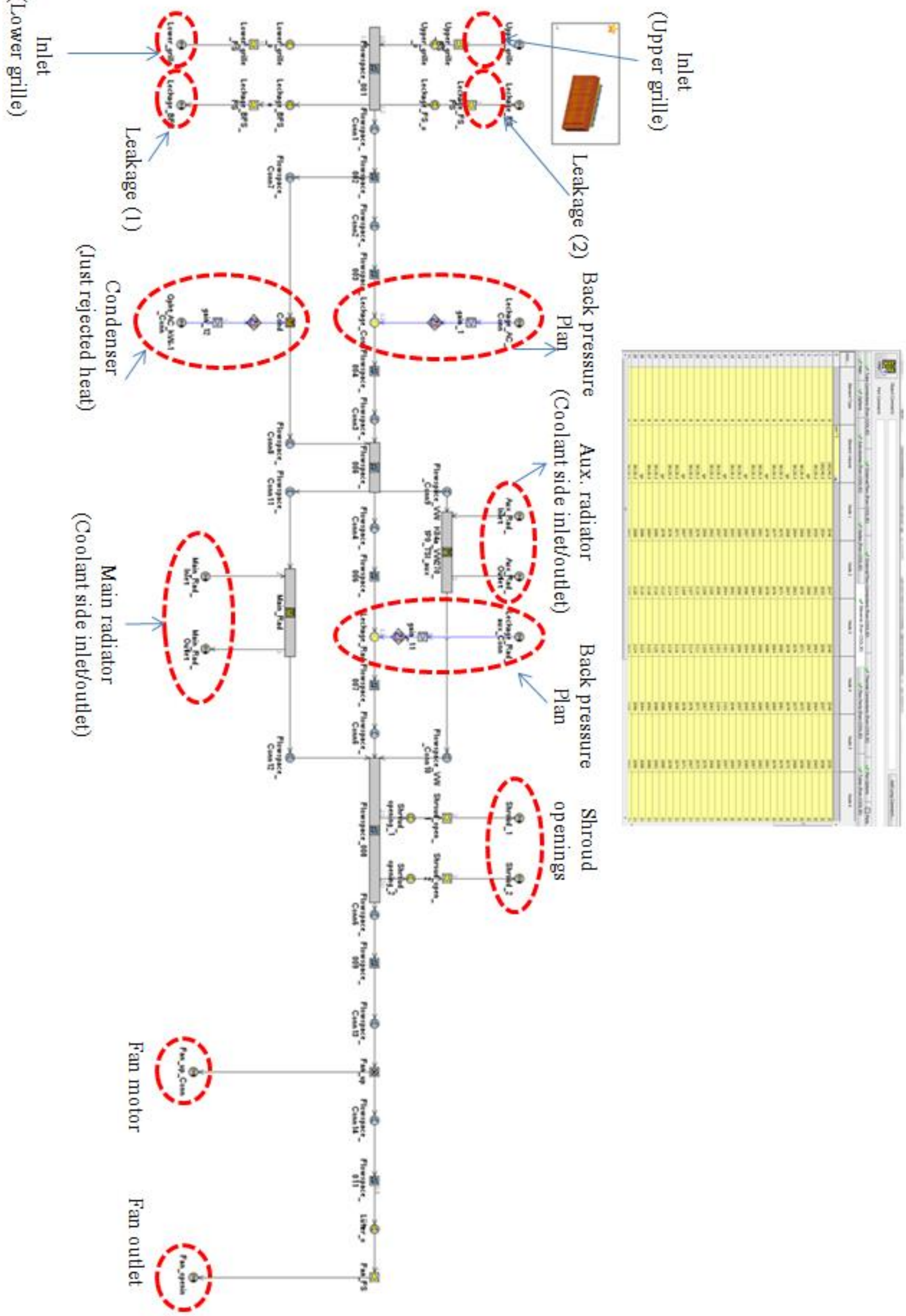

Figure 11. Discretization of the Underhood model

4. AMBIENT INTERFACES

The 1D block diagram (Fig. 11) is converted in a subassembly (block), as it is shown in the Fig. 12 highlighted in dashed red lines. This block is linked with ambient interfaces and the coolant side, as it is shown in the Fig. 12.

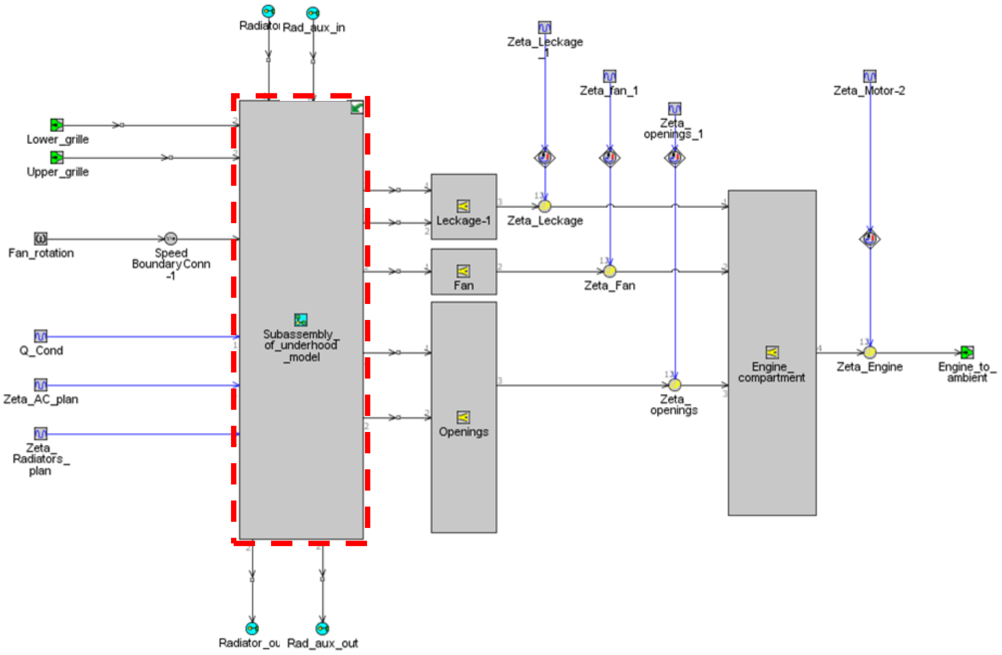

Figure 12. Interfaces with underhood model.

Each one of these interfaces has a correspondent block with physical data, as it is described in the Tab.2.

Table 2. Interfaces input data.

Interfaces	*Input data*	*Source*	*Variable*
Lower_grille	Static pressure	Desired condition	Constant
	Ambient Temperature	Desired condition	Constant
	Car velocity	Desired condition	Constant
	Drag coeffient (front vehicle)	CFD	Constant
Upper grille	Static pressure	Desired condition	Constant
	Ambient Temperature	Desired condition	Constant
	Car velocity	Desired condition	Constant
	Drag coeffient (front vehicle)	CFD	Constant
Qcon	Rejected heat	Experimental Data	Optimization
Zeta_AC_plan	Backpressure	Monitored via CFD	Optimization
Zeta_Radiators_plan	Backpressure	Monitored via CFD	Optimization
Fan_rotation	Fan rotation	Experimental Data	Constant
Leakage	Characteristic Lenght	Estimated	Constant
	Expantion Diameter	Estimated	Constant
Zeta_Leakage	Backpressure	Monitored via CFD	Optimization
Fan	Characteristic Lenght	Estimated	Constant
	Expantion Diameter	Estimated	Constant
Zeta_Fan	Backpressure	Monitored via CFD	Optimization
Oppening	Characteristic Lenght	Estimated	Constant
	Expantion Diameter	Estimated	Constant
Zeta_Oppening	Backpressure	Monitored via CFD	Optimization
Zeta_Engine	Backpressure	Monitored via CFD	Optimization
Ambient to Engine	Static pressure	Desired condition	Constant
	Ambient Temperature	Desired condition	Constant
	Car velocity	Desired condition	Constant
	Drag coeffient (front vehicle)	CFD	Constant

The 1D process of calculation should be monitored via CFD data; this interaction allows correcting the initial model, inputting values, which keep a similarity between the simulation 3D and 1D. In the Tab. 2, and column Variable, these variables are indicated as Optimization.

The expansion diameter of the components or connections: i) Leckage_1, ii) Fan, iii) Openings and iv) Engine compartment, should be estimated via hydraulic diameter (D_H) (eq. 6)

$$D_H = \frac{4.A}{P} \tag{6}$$

Where: A is the area of air passage and P is the perimeter around the air passage.

Although it is not object of the present analyze, it is important to say that the rejected heat from the engine and its intercooler system is inserted via others subassemblies connected in the interfaces Radiator_in and out, for the main cooling system, and Rad_aux_in and out for intercooler system. These two subassemblies systems have modeled pump maps and its properties, thermostatic valves, expansion tank of coolant and hoses, characterizing the coolant side for the calculation. The present work does not assess the coolant side.

5. COMPARISON BETWEEN RESULTS

Both analyses (CFD and 1D) were performed in steady state condition, and the target was to reproduce the effects of air flux passing through the radiators, which affects the temperature of stabilization on the cooling system.

These effects can be seen in the Fig. 13 (a) and (b), where it is shown the scalar field of air velocity on the radiators inlet using CFD (a) and 1D (b) graphical resources.

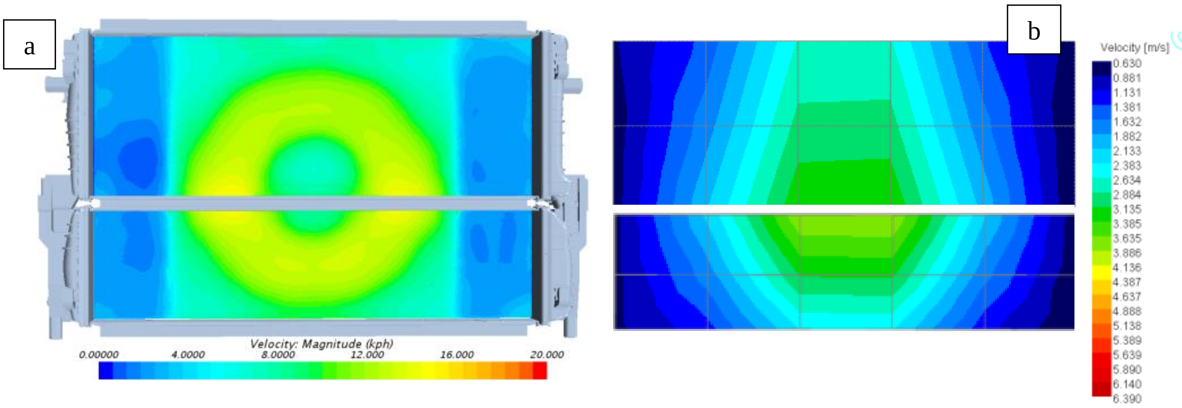

Figure 13. (a) CFD – scalar velocity representation, (b) 1D - scalar velocity representation.

It is noticed similarity between the air flux effects which was obtained in both methods of simulation.

The Tab. 3 shows comparative results between these methods of CFD and 1D analysis, for a given car and its cooling package.

Table 3. Comparison between CFD and 1D.

conditions	*Main Radiator*		*Aux Radiator*	
km/h	kg/s		kg/s	
	1D	CDF	1D	CDF
30	0.258	0.253	0.356	0.350
55	0.364	0.362	0.488	0.484
181	1.187	1.183	1.437	1.433

At least, it is possible to estimate the values of temperature of cooling system stabilization, when the same effects are obtained on the heat exchangers.

6. CONCLUSION

Although CFD simulation is rather complex, as it is seen in the Fig. 14, the present work proposes a productive interaction between 1D and 3D simulation focusing its analysis on air flux.

CFD approach aims to perform fluid mechanics interaction, with 12.10^6 cells, in comparison with 2434 cells used on 1D thermal analysis.

This methodology reduces the processing time, resulting on faster and attractive answers of cooling system studies.

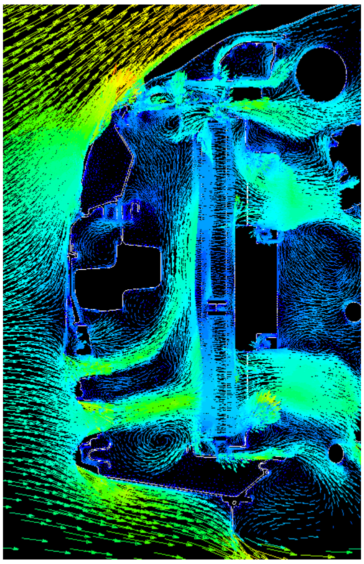

Figure 14. Vector Field of air flux passing through the heat exchangers

The same effect of air flux passing through the heat exchangers is obtained on 1D approach which assures accuracy on thermal analysis.

Without CFD analysis, it is impossible to forecast what is occurring on the heat exchangers and the thermal answers could be affected.

The values of air mass flow rate obtained via CFD are other important factor to check in order to analyze if the model is physically coherent.

In order to avoid prototypes and experimental tests, which reduces the cost of development, the present proposal of simulation has relevant contribution on predictive studies of cooling system.

7. REFERENCES

Abiodun Matthew Amao, Aug. 2007. *Mathematical model for Darcy Forchheimer flow with applications to well performance analysis*. Master of Science Thesis. Texas Tech University. Texas, USA.

Kanefsky P., Nelson V. A., Ranger M.. Nov. 1999. “A Systems Engineering Approach to Engine Cooling Design”. The 44th L. Ray Buckendale Lecture – SAE SP-154. Warrendale, USA.

Lisa L., Helena M., 2009. *Aerodynamic Drag Reduction of a Heavy Truck with Variable Cooling Air Intake Area*. Master’s Thesis in the Master’s program Automotive Engineering. Chalmers University of Technology, Department of Applied Mechanics, Division of Vehicle Safety. Göteborg, Sweden.

Maliska, Clovis R.. 2014. *Transferência de Calor eMecânica dos Fluidos Computacional*. LTC. Rio de Janeiro. 2nd edition.

Mouffouk, M. A.. 2014. *Aerodynamics Development of the Seguracing F1-R01 Prototype Using CFD*. University H.L of Batna. Department of Mechanical Engineering Specialty Aeronautical Engineering. Batna, Algeria.

Quim N.. 2007. *Desenvolvimento de Uma Metodologia de Simulação Aplicada ao Sistema de Arrecimento Veicular*. Master’s Thesis. Department of Mechanical Engineering, Universidade de São Paulo – USP. São Paulo – SP, Brazil.

Vdovinc, A.. 2010. *Cooling performance simulations in GT-Suite. 2010.* Master’s Thesis. *Department of Applied Mechanics Division of Vehicle Engineering & Autonomous Systems*. Chalmers University of Technology, Göteborg, Sweden.

8. RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.