XXVII Congresso Nacional de Estudantes de Engenharia Mecânica
08 a 12 de fevereiro de 2021, Curitiba, PR, Brasil

ON THE INVESTIGATION OF NUMERICAL METHODS FOR MODELLING THERMAL MANAGEMENT OF BATTERIES FOR AVIATION TO XXVII CREEM

Bianca Taís Visoná Carnielo, biancatcarnielo@gmail.com[1]
Carlos Roberto Ilário da Silva, carlos.ilario@embraer.com.br[2]
Aluisio Viais Pantaleão, aluisio.pantaleao@unesp.br[1]

[1]São Paulo State University (UNESP), School of Engineering Ilha Solteira. Avenida Brasil Sul, 56 Centro, Ilha Solteira-SP, 15385-000
[2]EMBRAER S.A, Avenida Brigadeiro Faria Lima, 2170, São José dos Campos-SP, 12227-901,

Resumo. *A indústria aeronáutica tem concentrado seus esforços em pesquisas e melhorias no contexto da eletrificação a fim de resolver problemas como emissão de gases, barulho e eficiência. Há muitos desafios a serem superados quando se trata de eletrificação de aeronaves, mas o um dos que merecem maior atenção é o gerenciamento térmico do sistema de baterias, devido à quantidade energia armazenada nos módulos e as exigências de segurança. Com o objetivo de melhorar o gerenciamento térmico de baterias para aviação, este artigo estuda um trocador de calor do tipo cold plate, no qual diferentes vazões e diâmetro dos tubos foram simulados a fim de compreender a influência dessas alterações na transferência de calor. Para isso, foram usados tanto o método numérico quanto analítico. O método analítico foi desenvolvido utilizando o software livre Octave, no qual equações que regem a transferência de calor foram implementadas, o método numérico foi desenvolvido utilizando a licença acadêmica do Fluent (Ansys). Até o momento, os resultados obtidos mostram que com um aumento na vazão a variação de temperatura do líquido na saída diminui, e com um aumento do diâmetro dos tubos essa variação de temperatura aumenta.*

Palavras chave: *Bateria. Aviação. Cold Plate. Fluent. CFD*

Abstract. *The aeronautic industry has concentrated its efforts on researches and improvements in the context of electrification aiming to solve problems like noise, gases emission and efficiency. There are many challenges to be tackled within an electrified aircraft but one that deserves special attention is the battery system thermal management, due to the amount of energy storaged in the modules and safety requirements. In order to improve thermal management of batteries for aviation, this paper studies a heat exchanger, in this case a cold-plate, where different flow rates and tube diameters were simulated aiming to understand the influence of these changes in heat transfer. For this porpouse, it was used both analytical and numerical methods. The analytical method was developed using the open source simulation software Octave, where heat transfer equations were implemented and the numerical method was developed using Fluent-Ansys academic license. So far, the results obtained have shown that with an increase in flow rate the temperature variation decreases, and with an increase in the tube diameter the temperature variation also increases..*

Keywords:*battery, aviation, cold-plate, Fluent, CFD*

Nomenclature

Greek letters

ϕ	Key variable
ρ	Specific mass [kg/m^3]

Symbols

ΔT	Temperature Variation [°C]
A_b	Cross sectional area of cold plate base [m^2]
A_s	Tube surface area [m^2]
D	Tube diameter [mm]
d	Tube diameter [m]
E	Battery voltage [V]
e_a	Approximate relative error
e_{ext}	Extrapolated relative error
GCI	Grid convergence index
h	Heat transfer coefficient

I	Battery Current [A]
k	Aluminum thermal conductivity [W/m.K]
k_f	Fluid thermal conductivity [W/m.K]
N	Mesh number of elements
Nu	Nusselt Number
p	Apparent order
Pr	Prandtl Number
Q	Heat [W]
r	Grid refinement factor
R_{cond}	Conduction thermal resistence
R_{conv}	Convection thermal resistence
R_t	Total thermal resistence
Re	Reynolds Number
t	Thickness
V	Portion of power effectively used [V]

Subscript

a	Approximate
$cond$	Conduction
$conv$	Convection
ext	Extrapolated
t	Total

1. INTRODUCTION

Electrification applied in aircrafts is a solution for issues like gases emission, efficiency and noise. According to Cao *et al.* (2012), it is estimated that an AEA (All Electric Aircraft) can reduce aircraft weight by 10% and fuel consumption by 9%. An electrified aircraft can be classified into three levels: More Electric Aircraft (MEA), All Electric Aircraft (AEA) and Hybrid Electric Vehicles (HEV). The MEA has the goal of replacing mechanical, hydraulic and pneumatic system with electrical equipment, the AEA aims to replace fuel propulsion with all electrical propulsion, and in the case of HEV one or several electric motors are installed to participate in the propulsion force with the Internal Combustion Engine (Zhang *et al.*, 2008).

Although electrical propulsion has its benefits, there are some technologycal barriers that need to be solved, mainly about battery performance, re-charging speed and safety.

High temperatures can harm battery cycle of life and performance, it also means a risk to the electric system. Thermal management is very necessary because safety is one of the most important requirements in aviation (Berger, 2017). This paper state of contribution is to help electrification to be safer and easier to implement.

Gas emission is a relevant necessary and the main goal of aircrafts electrification, data from the Salvador International Airport shows that in the period of one year it is emitted the amount of gases in the list below, that could be reduced: (Braga and de Albuquerque, 2015)

- $1,8 \times 10^8$ kg of CO_2;
- $5,3 \times 10^5$ kg of NO_x;
- $4,1 \times 10^5$ kg of CO;
- $4,1 \times 10^4$ kg of SO_2;
- $4,7 \times 10^3$ kg of NO_2;
- $4,1 \times 10^3$ kg of CH_4.

To keep battery pack at an optmum average temperature it is used heat exchangers, they can work with air cooling or liquid cooling. Due to better efficiency of heat transfer by liquid cooling, it was selected to deepen its studies in this paper.

In this case, it was used a cold plate as heat exchanger, which consists of a metal tiny wall with several liquid channels. The liquid can be water, water-glycol, refrigerants or oil, here we considered the use of water (Huo *et al.*, 2014). The goal is to understand how changes in mass flow rate and tubes diameter can influence heat exchange. Two different methods were implemented and the results compared in this paper. First, an analytical method using the open source simulation software Octave was used and then, a numerical model was developed and simulated using the CFD software Fluent by Ansys, with academic license (Ansys, 2020).

The model of the batteries studied in this work is the Lithium-Iron-Phosphate ($LiFePO_4$), according to Niculuta and Veje (2012) they are one of the most preferred choices of energy storage for electric vehicles, because they provide high voltages, more capacity and cycle of life at reduced weight. Some considerations are necessary for this model: heat transfer inside the battery pack can be neglected because of electrolytes fluid little mobility and heat transfer rate through radiation is also neglected (Niculuta and Veje, 2012). Some specific informations about the battery are presented in Tab. 1.

Table 1. Informations about battery geometry and power

Property	Value
Length	115 mm
Width	61 mm
Height	203 mm
Voltage	3.2 V
Current	70 A

2. METHODOLOGY

Two values for tube diameter were studied: 5mm and 7mm, each one for different mass flow rate between 0.02kg/s and 0.08kg/s. Both analytical and numerical methods were applied. Figure 1 shows the design developed using Design Modeler with the boundaries condition imposed: heat flow of 27 kW/m^2 at the top surface and the other surfaces were insulated.

The cold plate length and width are the same of batterie's and the cold plate height it's 3 times the diameter.

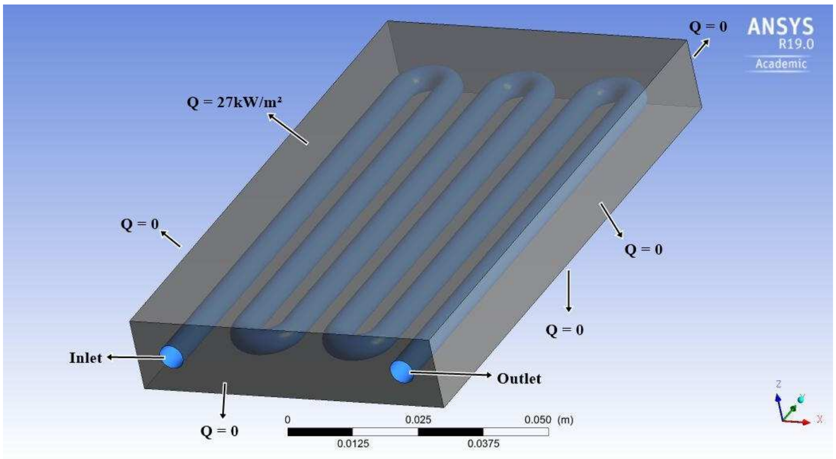


Figure 1. Cold plate geometry and boundaries conditions

2.1 Analytical Modelling

Using the software Octave, the equations according to Maddipati *et al.* (2013) were implemented in a routine to calculate the temperature variation in function of the volumetric flow rate for two values of tube diameter (5mm and 7mm). In order to know the temperature variation, it is necessary to calculate thermal resistances, for this, it was used Reynolds Number (Re) and Nusselt Number (Nu), which in this case, follows the relation below:

$$Nu = 0.023 \cdot Re^{0.8} \cdot Pr^{0.4} \tag{1}$$

Where Pr is the Prandtl number. Then, it is possible to find the heat transfer coefficient h.

$$h = \frac{Nu \cdot k_f}{d} \tag{2}$$

So, the total thermal resistance is:

$$R_t = R_{conv} + R_{cond} \tag{3}$$

Where R_{conv} is the thermal resistance due to convection and R_{cond} due to conduction.

$$R_{cond} = \frac{t}{k \cdot A_b} \tag{4}$$

$$R_{conv} = \frac{1}{h \cdot A_s} \tag{5}$$

Where A_b and A_s are cross sectional area of cold plate base and tube surface area, respectively.

Therefore, the temperature variation will be:

$$\Delta T = R_t \cdot Q \tag{6}$$

The heat Q is given by:

$$Q = I(E - V) \tag{7}$$

Where I is current, E is battery voltage and V is the portion of power that is effectively used to provide energy, in this case it was considered 10% of battery voltage (V).

2.2 Numerical Modelling

Aiming to know the results using a numerical method, the cold plate system was simulated in Fluent, using Academic License (Ansys, 2020). The geometry and mesh were also generated using Ansys softwares. Figure 2 shows the tube mesh zoom, where a tetrahedron mesh with prism layer inside the tubes was used.

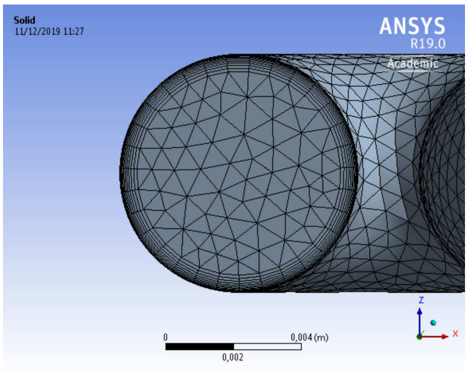


Figure 2. Tube mesh zoom

3. RESULTS

The results for analytical and numerical methods are disposed below.

3.1 Analytical Validation

In order to validate the equations used in the analytical method, the temperature in function of volumetric flow rate, was calculated in another condition, the reference values are from Maddipati *et al.* (2013). The results obtained are presented in Fig.3 that shows a comparison between calculated and literature data (Maddipati *et al.*, 2013). Although there is a difference between the curves, the results obtained are considered satisfactory as the error in the worst condition is 3.6%. So, the analytical method can be used as a reference to CFD analysis.

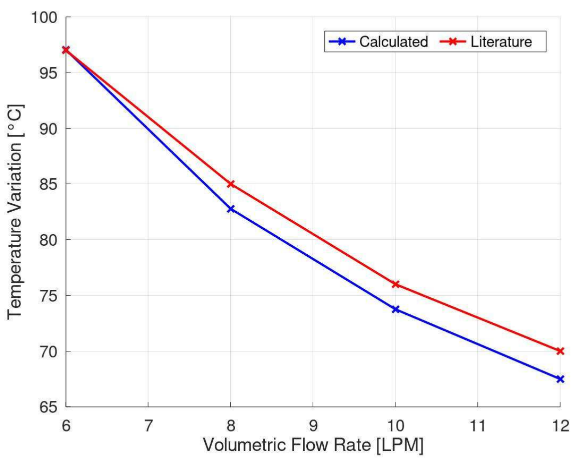


Figure 3. Cold Plate temperature in function of volumetric flow rate calculated and from literature

Since the analytical method was validated using literature data (Maddipati *et al.*, 2013), it can be used as a reference to CFD analysis, even if all fluid flow and heat transfer phenomenology are not included.

3.2 Analytical Results

According to the equations on Section 2.1, the temperature variation for different cases are those shown in Tab. 2 for tube with diameter of 5mm and Tab. 3 shows the results for 7mm tube diameter.

Table 2. Analytical results for D = 5mm

Flow Rate [kg/s]	Temperature Variation [$°C$]	Reynolds
0.02	3.07	5068
0.04	2.49	10137
0.06	2.27	15205
0.08	2.16	20274

Table 3. Analytical results for D = 7mm

Flow Rate [kg/s]	Temperature Variation [$°C$]	Reynolds
0.02	4.20	3620
0.04	3.14	7240
0.06	2.74	10861
0.08	2.53	14481

3.3 Mesh Analysis

In this section are the mesh sensitivity study and y+ analysis, so the considerations made for numerical method can be validated.

3.3.1 Mesh Sensitivity

The mesh sensitivity study was made by Grid Convergence Method (GCI) according to the procedure in Celik *et al.* (2008), the study was made considering just the tubes mesh.

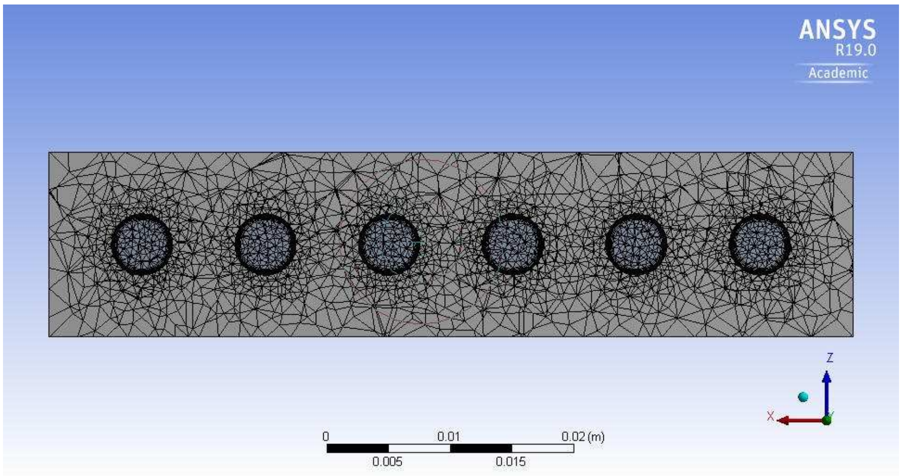
Three meshes were selected, the outlet temperature was considered as a key variable (ϕ), because it is important for the objective of this simulation. The results are in Tab. 4.

Table 4. Mesh Analysis

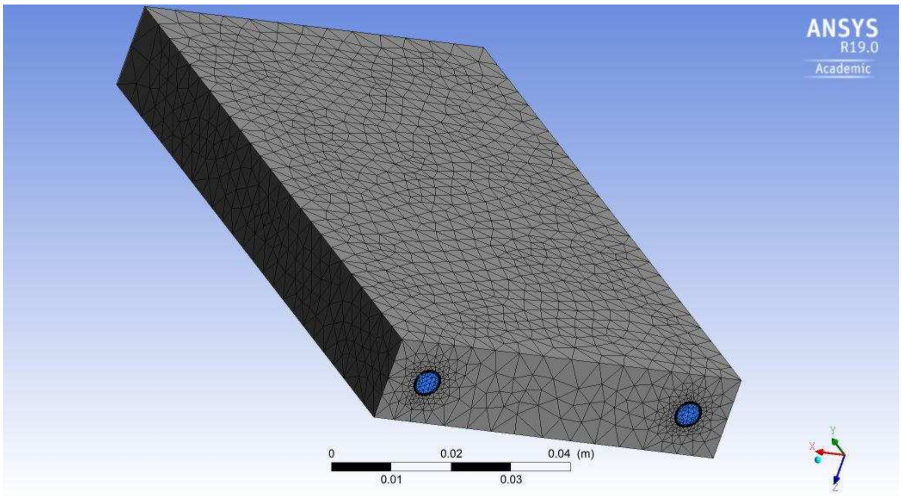
Parameter	Value
N1, N2, N3	2616840, 691386, 302616
r_{21}	1.55
r_{32}	1.31
ϕ_1	302.5463 K
ϕ_2	302.6144 K
ϕ_3	302.6935 K
p	1.80
ϕ_{ext}^{21}	29.49
e_a^{21}	0.0023
e_{ext}^{21}	0.0019
GCI_{fine}^{21}	0.24%
GCI_{coarse}^{32}	0.53%

Where, N is the number of elements of the mesh (just for the tubes), r is the grid refinement factor that must be higher than 1.3 and GCI is the grid convergence index, it is expected GCI be lower than 2.0%. So, the mesh used for this simulation is satisfactory.

The Figure 4 shows details of the mesh inside and outside the cold plate.



(a) Mesh inside the cold plate



(b) Mesh outside the coldplate

Figure 4. Mesh Details

3.3.2 y plus

The wall y+ is a non-dimensional distance similar to local Reynolds number, it is the ratio between the turbulent and laminar influences in a cell (Salim and Cheah, 2009).

This parameter is used to certify that turbulent model was correctly chosen, it must be between 1 and 7. The Figure 5 shows the y+ contours, in this case, the range was between 2 and 4.

The Reynolds Numbers in Tabs. 2 and 3 are higher than 2300, so the analytical and CFD methods converged about turbulent flow model.

3.4 Numerical Results

Figure 6 shows the temperature variation in function of flow rate obtained using analytical and numerical methods for both tube diameters, it is possible to see that for a smaller diameter the variation is lower. And with the increase in flow rate, the temperature variation decreases.

The massic flow rate depends on specific mass, velocity and area ($\dot{m} = \rho V A$), so if there is an increase on tube diameter, for the same flow rate, the fluid flow velocity must decrease, and as a consequence the heat transfer is lower, because the heat transfer coefficient (h) depends on Reynolds Number that decreases with the velocity.

There is a difference between CFD simulation and analytical method. This happens because the analytical method implemented does not consider all the phenomenology associated with fluid flow and heat transfer, like laminar or turbulent flow, 3D heat transfer and pressure drop, for example.

Turbulent flows are more efficient to thermal management, also is 3D heat transfer, once the CFD analysis takes this into account, the results for temperature variation are lower using this methodology.

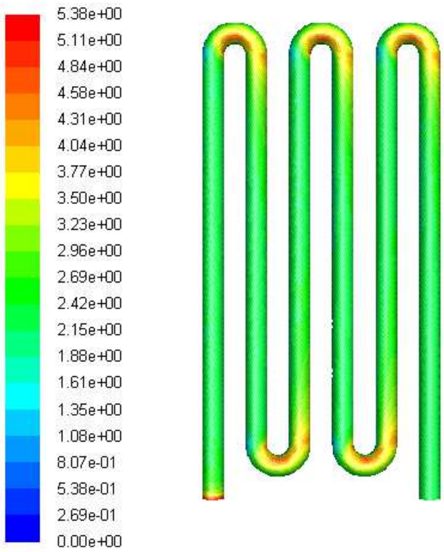


Figure 5. y+ contours for tube diameter 5mm and flow rate 0.08kg/s

Though CFD and analytical absolute values are different, they show the same behavior and the curves inflection happens at the same point.

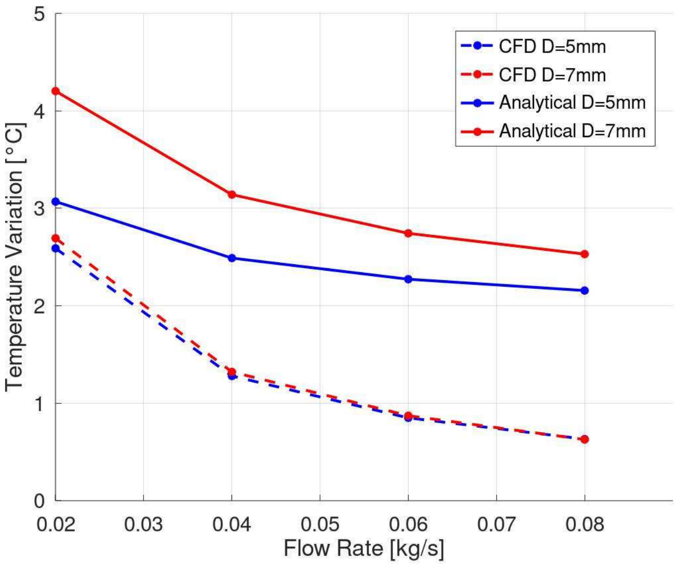


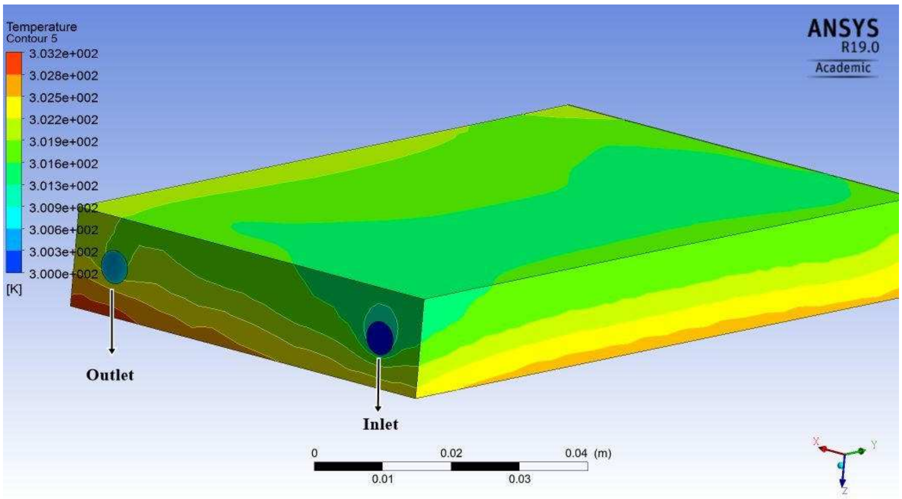
Figure 6. Temperature variation in function of flow rate

Figure 7 shows cold plate temperature contours and tubes temperature contours obtained at Fluent for a diameter of 5mm and flow rate of 0.04kg/s.

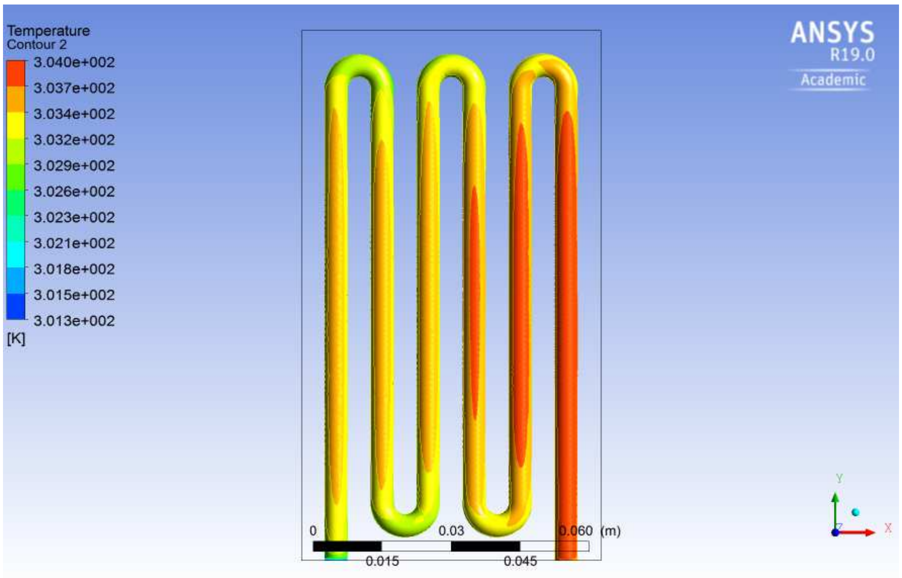
In this case, the pressure drop is caused by fluid friction with the tube walls, the expression for this is: (Fox and McDonald, 2001)

$$\Delta h = f \frac{L \cdot v^2}{D \cdot 2g} \tag{8}$$

According to the equation above, the pressure drop increases with the velocity and decreases with tube diameter, so the results disposed in Fig. 8 are physically expected.



(a) Cold plate



(b) Tubes

Figure 7. Temperature contours for D=5mm and flow rate 0.02kg/s.

4. CONCLUSIONS

Both anylitical and CFD methods showed that the use of a smaller tube diameter improved the heat transfer, however this means an increase in pressure drop. The same happens with an increase in flow rate.

In this case, the heat transfer depends basically on flow velocity, a smaller tube increases the velocity and, as consequence, the efficiency of the heat exchanger. On the other hand, the fluid friction with the tube walls is higher with a smaller tube and also increases with flow rate, that explains the pressure drop behavior.

The lower values for temperature variation for CFD analysis compared to analytical is a consequence of the methodology which takes into account the turbulent flow and 3D heat transfer, since this features improve thermal management efficiency. So far, the study showed that the results obtained using both analytical method and CFD have the same behavior, but a difference in absolute values.

The mesh sensitivity study validated the simulatios, once the difference between fine and medium mesh is about 0.24%, using the outlet temperature as key variable. The difference between medium and coarse mesh also shows a satisfactory result of 0.53%.

This work is the beginning of a new research line for Ilha Solteira academic community and the results obtained here will guide future efforts on this topic so important for aircrafts eletricfication, once batteries thermal management is necessary to improve aircrafts safety and efficiency.

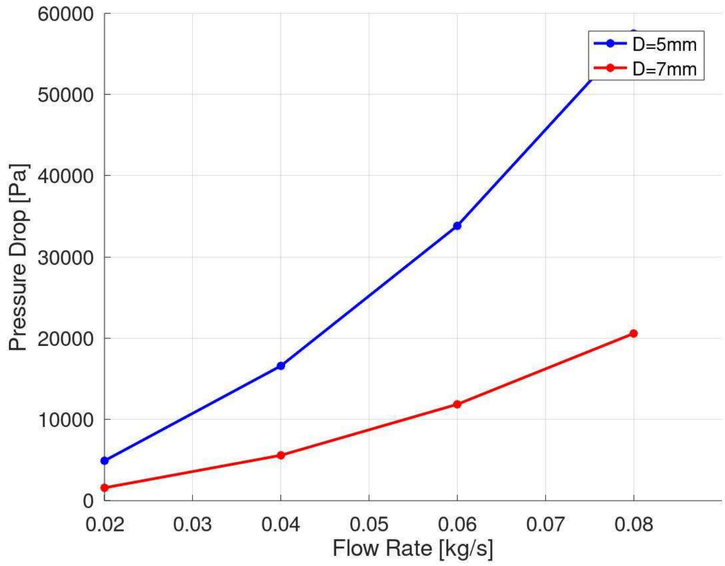


Figure 8. Pressure drop in function of flow rate

5. ACKNOWLEDGEMENTS

To Embraer for the opportunity and financial support and UNESP Ilha Solteira for support and structure, especially FEPISA.

This work has been possible in function of the São Paulo Research Foundation (FAPESP) grants, processes 2019/07947-0 (Regular Process).

6. REFERENCES

Ansys, 2020. *Fluent - Fluid Simulation Software*. URL `https://www.ansys.com/products/fluids/ansys-fluent`. Last access: 2020-07-21.

Berger, R., 2017. “Aircraft electrical propulsion - the next chapter of aviation”.

Braga, T.C. and de Albuquerque, E.L., 2015. “Emissoẽs de poluentes provenientes do trafego de aeronaves no aeroporto internacional de salvador.” In *XI Congresso Brasileiro de Engenharia Quimica em Iniciação Cientifica*.

Cao, W., Mecrow, B.C. *et al.*, 2012. “Overview of electric motor technologies used for more electric aircraft (mea)”. *IEEE Transactions on Industrial Electronics*, Vol. 59, No. 9.

Celik, I.B., Ghia, U., Roache, P.J., Freitas, C.J., Coleman, H. and Raad, P.E., 2008. “Procedure for estimation and reporting of uncertainty due to discretization in cfd applications”. *Journal od Fluids Engineering*.

Fox, R.W. and McDonald, A.T., 2001. *Instroduction to Fluid Mechanics*. John Wiley and Sons, Inc.

Huo, Y., Hao, Z., Liu, X. and Zhao, J., 2014. “Investigation of power battery thermal management by using mini-chanel cold plate”. *Elsevier*.

Maddipati, U.R., Rajendran, P. and Laxminarayana, P., 2013. “Thermal design and analysis of cold plate with various proportions of ethyl glycol water solutions”. *International Journal of Advanced Trends in Computer Science and Engineering*, Vol. Vol.2, No. No. 6, pp. Pages 22–25.

Niculuta, M.C. and Veje, C., 2012. “Analysis of the thermal behavior of a lifepo4 battery cell”. *Journal of Physics*.

Salim, M. and Cheah, S., 2009. “Wall y+ strategy for dealing with wall-bounded turbulent flows”. *Proceedings of the International MultiConference of Engineers and Computer Scientists 2009 Vol II*.

Zhang, H., Saudemont, C., Robyns, B. and Petit, M., 2008. “Comparison of technical features between a more electric aircraft and a hybrid electric vehicle”. In *2008 IEEE Vehicle Power and Propulsion Conference*. pp. 1–6. ISSN 1938-8756. doi:10.1109/VPPC.2008.4677663.

7. RESPONSIBILITY NOTICE

The authors are solely responsible for the printed material included in this paper.