DESIGN AND ANALYSIS OF AN EMBEDDED DUAL CHANNEL COOLING STRUCTURE FOR AN AXIAL FLUX PERMANENT MAGNET SYNCHRONOUS MOTOR

Flavia Coli,[1] Erica Iozzia,[1] Matteo Fasano,[1] Jonathan Hey[2,*]

[1]Department of Energy, Politecnico di Torino, Corso Duca degli Abruzzi, 24, 10129 Torino, Italy
[2]Singapore Institute of Manufacturing Technology (SIMTech), Agency for Science, Technology and Research (A*STAR), 2 Fusionopolis Way, #08-04 Innovis, Singapore 138634, Republic of Singapore

ABSTRACT

This study focuses on the thermal management of an Axial Flux Permanent Magnet Synchronous Motor (AFPMSM) designed for an electric two-wheeler. The aim of the analysis is to enhance the performance of the motor through an effective heat removal strategy. The temperature rise in the motor affects the overall efficiency and performance of the machine. Effective thermal management allows an increase of the torque-speed limit as well as the power output while maintaining the same motor's physical dimension. In particular, the PMSM under investigation has a central stator structure with embedded cooling channels within the stator to better exploit the available space. In this paper, several configurations of the cooling channels are analysed to minimize the motor's winding temperature rise. A dual channel cross-flow configuration is considered in order to improve the temperature uniformity on the stator and windings. The channel design is targeted at enhancing the effectiveness of the heat dissipation by introducing secondary flow, i.e. turbulence. This is mainly achieved with a curved channel geometry. A conjugate heat transfer analysis is conducted using Finite Element Analysis (FEA) to model the temperature distribution. Specifically, the conductive heat transfer in the solid domains and the convective one in the fluid domain are investigated. A comparison between the proposed cooling channels is performed by adopting a dimensionless Performance Evaluation Criterion (PEC). A value greater than 1 indicates an improvement in the cooling design with respect to a straight pipe configuration. This metric considers both temperature and pressure related aspects, providing an evaluation of the best overall performance.

KEY WORDS: AFPMSM, thermal management, cooling channels, dual channel configuration, conjugate heat transfer, FEA

NOMENCLATURE

A	Cross section area	(m^2)	PEC	Performance evaluation criterion	(-)
A_c	Contact area	(m^2)	ΔP	Pressure drop per unit length	$(\frac{Pa}{m})$
D_h	Hydraulic diameter	(m)	Pr	Prandtl number	(-)
f	Friction factor	(-)	Q	Motor's losses	(W)
h	Convective heat transfer coefficient	$(\frac{W}{m^2 K})$	Re	Reynolds number	(-)
k	Water thermal conductivity	$(\frac{W}{m\ K})$	R_{th}	Thermal resistance	$(\frac{K}{W})$
μ	Water dynamic viscosity	$(Pa\ s)$	ρ	Water density	$(\frac{kg}{m^3})$
Nu	Nusselt number	(-)	T_f	Average fluid temperature	$(°C)$
P	Wet perimeter	(m)	T_s	Channel surface temperature	$(°C)$

*Corresponding Jonathan Hey: jonathan-hey@simtech.a-star.edu.sg

u_τ	Friction velocity	$(\frac{m}{s})$	y	Near-wall distance	(m)
v	Water inlet velocity	$(\frac{m}{s})$	y^+	Dimensionless wall distance	(-)
$\dot{V}$	Volumetric flow rate	$(\frac{l}{min})$			

1. INTRODUCTION

Thermal management of electric motors is crucial as the automotive industry shifts towards more electrically driven vehicle propulsion systems. Managing the heat generated from both power electronics and electric motors becomes more challenging as the industry shifts towards smaller components size and reduce weight while maintaining performance and reliability [1]. An optimized thermal design can substantially increase the rated power of a machine without significantly increasing the physical size [2, 3].
Heat transfer in electric motors is complex due to the multiple paths for heat transfer and different materials involved. The multiple layers of materials and many interfaces limit heat transfer in such machines [4]. This can create hot spots in the machine such as the electrical windings and reduce motor performance over time. The chosen cooling method affects how effective heat flow is and the final temperature distribution [5].
Compared to conventional radial motors, Axial Flux Permanent Magnet Synchronous Motors (AFPMSMs) have higher torque and higher power density. They are lighter and more compact, making them suitable for electrical vehicles [6, 7]. However, one of the main drawbacks in such a design is the accessibility of the stator and winding assembly for active cooling with liquids. Moreover, the higher power density and the enclosed design can lead to a significant heat-generation that affects the overall motor's performance [6]. Indirect cooling, through cooling jacket in the motor housing is a common approach. In particular, the jackets are generally located in the outer diameter (OD) of the motor for a single-stator double-rotor configurations, and within the end cover for double-stator and single-rotor configurations. This increases the thermal resistance between the windings and the coolant, leading to a lower heat dissipation [8].
In this paper, an embedded cooling channel configuration in the stator core is proposed to overcome this problem. It is positioned closer to heat-generating components, reducing the thermal resistance of the heat transfer path. In this way, it is possible to have a compact design while maximising the available space and the cooling effectiveness. A comparative analysis between three cooling channel designs is conducted with the aim of selecting the one that leads to the best heat transfer while minimizing the pressure drop.

2. COOLING CHANNEL DESIGN

2.1 Cooling configurations

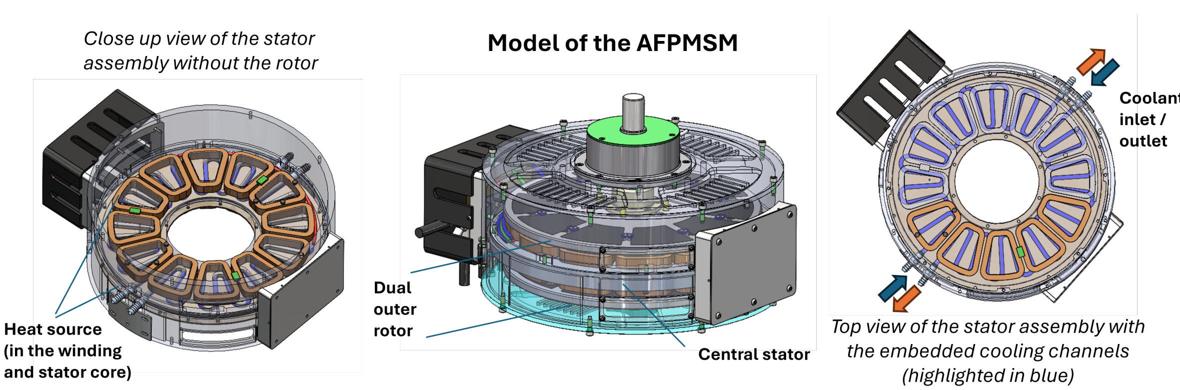


Fig. 1 Model of the AFPMSM machine and illustration of the embedded cooling channels.

The motor under analysis is an Axial Flux Permanent Magnet Synchronous Motor (AFPMSM) with a Single Stator Double Rotor (SSDR) structure. The cooling channel is directly embedded in the stator as shown in Figure 1. This allows for a better utilization of the available volume, making it more compact and further

enhancing the power density. The stator has an outer diameter of 0.33 m with an inner diameter of 0.13 m and a total thickness of 0.05 m.
In order to exploit the whole stator volume, a cooling channel configuration with two separate and identical channels is designed. It is characterized by two inlets and two outlets, with a counter flow configuration in the two channels. This solution is adopted to minimize the temperature gradient within the stator structure. The cross section area is rectangular with an height of 10 mm and a width of 6 mm. The cooling fluid is water and the selected volumetric flow rate is 5 $\frac{l}{min}$. To establish the flow field the Reynolds number is evaluated:

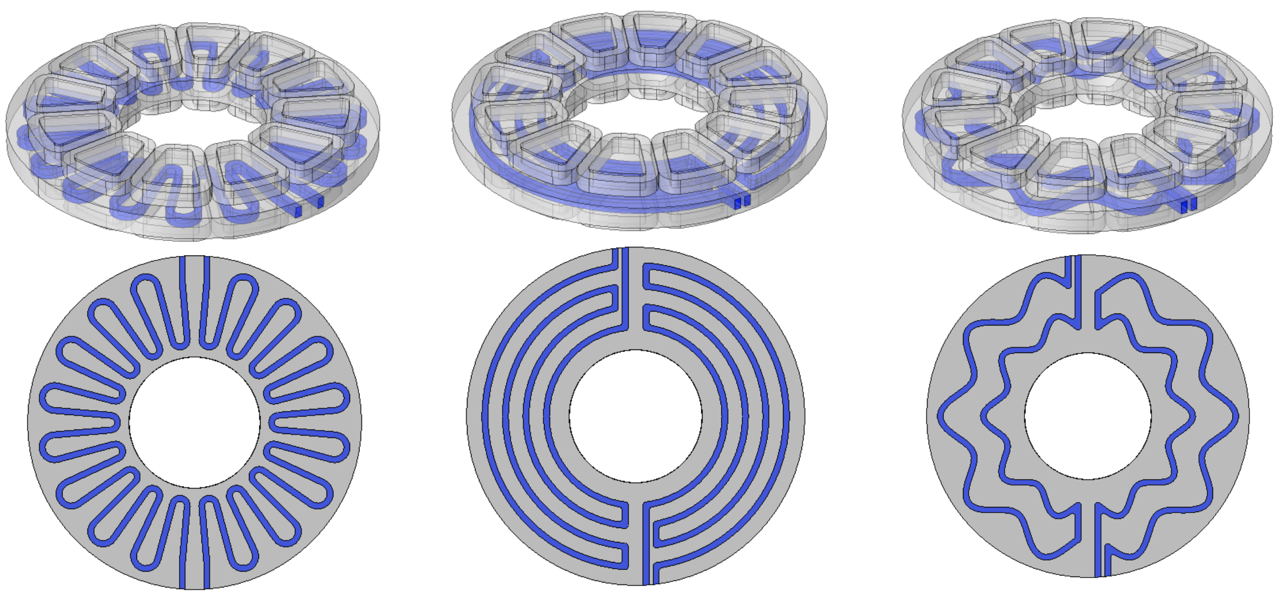
$$Re = \frac{\rho\, v\, D_h}{\mu}, \tag{1}$$

where the inlet velocity and the hydraulic diameter are calculated respectively as

$$v = \frac{\dot{V}}{A} \tag{2}$$

$$D_h = \frac{4\,A}{P}. \tag{3}$$

As the Reynolds number is higher than 2300, the flow is expected to be turbulent. A serpentine, a tandem and wavy channel configuration are proposed for this study, as shown in Figure 2. The external dimensions of the stator assembly are unchanged and the flow rate are kept the same for all three designs.



(a) (b) (c)

Fig. 2 (a) Serpentine, (b) Tandem, (c) Wavy configurations for the simulated heat exchanger.

2.2 Model set-up

A steady-state study is carried out through a conjugate heat transfer analysis using COMSOL software. A simplified model of the stator assembly is considered. Specifically, it consists of three domains: windings, stator and fluid domain. The windings are made of copper and they represent the heat source of the system. The material selected for the stator is AISI 1018. The cooling channel system is modelled by the fluid domain. The Navier-Stokes equations and the Fourier model are used to model both the conduction and the convection in the solid and fluid domains. The heat generated as result of the electrical losses during the operation is estimated at a 1000 W for this particular model. To model it, a volumetric heat generation boundary condition is applied over the coils domain. This assumption simplifies the analysis, while it provides a reasonable distribution of the heat source. The inlet fluid velocity and the inlet fluid temperature are respectively equal to 1.4 $\frac{m}{s}$ and 20 $°C$ and they are assumed to be constant at the inlet. Based on the no-slip condition, the wall fluid velocity is zero. The pressure at the outlet is specified as atmospheric pressure. The whole system is well-insulated, therefore it does not exchange a significant amount of heat with the external environment (i.e. adiabatic assumption). Consequently, the only heat sink for the dissipated thermal energy is the fluid domain. The Low Re k-ε turbulence model is adopted to study the flow field, as it allows to accurately solve the viscous sub-layer, leading to a more precise prediction of near-wall mass and heat transfer. A key parameter to accurately investigate the boundary layer is the wall distance, namely:

$$y^+ = y\,\frac{\rho\, u_\tau}{\mu}. \tag{4}$$

A value of y^+ lower than 5 ensures that the viscous sub-layer is well represented [9]. To meet this requirement, the near-wall distance is adjusted by refining the mesh. All the analysis is conducted considering only half of the entire system, taking advantage of the central symmetric stator structure. Thus, the computational cost is significantly reduced.

3. SIMULATION RESULTS

In this section, the simulation results are presented and analysed to determine the optimal cooling configuration, considering both thermal performance and pumping energy consumption. In particular, three different performance metrics are employed: the thermal resistance, the pressure drop and a dimensionless Performance Evaluation Criterion (PEC).

The thermal resistance is used to evaluate the thermal performance of the cooling systems. It takes into account the convective heat transfer coefficient and the contact area. Lower thermal resistance indicates more efficient heat dissipation. It is defined as:

$$R_{th} = \frac{1}{h\,A_c}, \tag{5}$$

where the heat transfer coefficient of the fluid is obtained from the Newton's law:

$$h = \frac{Q}{A_c\,(T_s - T_f)}. \tag{6}$$

This last parameter gives an insight of the cooling effectiveness of the system, considering the heat transferred to the fluid and the temperature difference between the average channel surface temperature and average fluid temperature. These temperatures are derived from FE simulations performed in COMSOL.

From Table 1, it can be observed that comparing the tandem channel and the serpentine one, even if they have a similar contact area, the first shows the lowest thermal resistance due to its higher heat transfer coefficient. The wavy configuration, despite having a heat transfer coefficient between the tandem and serpentine, presents the highest thermal resistance due to the limited surface area. The R_{th} values are coherent with the observed temperature reduction in the coils. Indeed, due to the low R_{th} value, the average temperature of the stator windings in tandem design is 23.1 $°C$, while the wavy design has an average temperature of 23.7 $°C$.

Configuration	h [$\frac{W}{m^2K}$]	A_c [m^2]	R_{th} [$\frac{K}{W}$]
Serpentine	14299	0.051	1.37×10^{-3}
Tandem	17794	0.048	1.17×10^{-3}
Wavy	16749	0.035	1.74×10^{-3}

Table 1 Heat transfer coefficient, heat exchange area and thermal resistance for the simulated heat exchanger configurations.

Another parameter analyzed is the pressure drop per unit length. It is a measure of the cooling system's energy efficiency, as it is directly related to the energy consumed by the pumping system. It is mainly affected by the channel's geometry at a given flow rate. The Table 2 summarizes the pressure drop for each configuration. The wavy configuration shows the best performance in terms of energy consumption, since it has a smoother flow path with respect to the other two designs. On the other hand, the serpentine channel has many sharp turns and so it exhibits the highest pressure drop.

Configuration	ΔP [$\frac{Pa}{m}$]
Serpentine	33205
Tandem	26923
Wavy	25493

Table 2 Pressure drop per unit length for the simulated heat exchanger configurations.

To evaluate the overall performance of the cooling strategies proposed, the dimensionless performance evaluation criteria (PEC) is used [10]. It accounts for both heat transfer and pressure losses, allowing a comparison of different geometries considering a straght pipe as reference case. A PEC value higher than 1 indicates that the improvement in the heat transfer is greater than the increase in pressure losses, making the configuration more efficient than the reference one. It is computed as:

$$PEC = \frac{\frac{Nu}{Nu_0}}{(\frac{f}{f_0})^{\frac{1}{3}}}, \tag{7}$$

where Nu_0 and f_0 are respectively the Nusselt number and the friction factor for the reference straight pipe. These are calculated according to the Gnielinski correlation:

$$Nu_0 = \frac{(\frac{f_0}{8})(Re - 1000)P_r}{1 + 12.7(\frac{f_0}{8})^{\frac{1}{2}}(P_r^{\frac{2}{3}} - 1)} \tag{8}$$

$$f_0 = (0.790\ ln\ Re - 1.64)^{-2}. \tag{9}$$

Nu and f for the case under analysis are instead calculated as:

$$Nu = \frac{h\ D_h}{k} \tag{10}$$

$$f = \frac{2\ \Delta P\ D_h}{\rho\ v^2}. \tag{11}$$

The results are shown in Table 3. The PEC is a balance between the thermal performance and the energy consumption. The tandem configuration has the highest value, indicating the best overall performance, as the heat transfer coefficient is high and the pressure drop is moderate. On the other hand, the serpentine configuration is penalized by the high pressure drop which also leads to a lower PEC value compared to the other two.

Configuration	PEC [-]
Serpentine	1.07
Tandem	1.43
Wavy	1.37

Table 3 Dimensionless performance evaluation criterion for the simulated heat exchanger configurations.

The temperature distribution of the motor with the tandem cooling channel design is shown in Figure 3. As expected, the highest temperature is observed in the windings where the heat is generated. This heat is transferred to the stator through conduction and absorbed by the coolant that acts as heat sink. The choice of dual channel allows a shorter length for each channel compared to a single one. As results, the flow path is shorter and this leads to a lower outlet fluid temperature, making the heat dissipation more effective. Moreover, the cross flow configuration ensures the temperature uniformity of the system preventing hot spots.

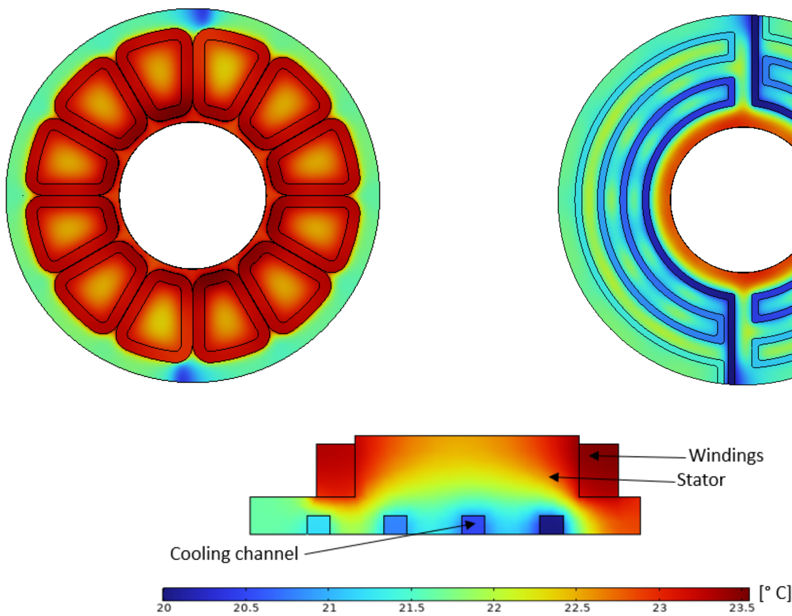


Fig. 3 Steady state temperature distribution of stator, windings and cooling channel in the tandem configuration.

4. CONCLUSIONS

In this paper a conjugate heat transfer analysis is carried out, focusing on the thermal management of a central stator double rotor axial flux permanent magnet synchronous motor. Through an effective heat removal strategy, it is possible to enhance the overall performance without changing the motor's size. Three different cooling channels configurations - serpentine, tandem and wavy - are designed. They are embedded within the stator structure to optimize the available space for the heat dissipation. A comparative analysis is developed to investigate the outcomes in terms of thermal effectiveness and pressure drop. Three performance metrics are chosen: thermal resistance, pressure drop and a dimensionless performance evaluation criterion (PEC). To evaluate the thermal behaviour, the thermal resistance parameter is considered. The tandem channel configuration shows the best result due to its higher heat transfer coefficient. To assess the energy efficiency, the pressure drop is used, as it is directly related to energy required by the pumping system. The wavy channel exhibits the best performance thanks to its smoother path. Finally, to determine the most effective cooling configuration, a dimensionless index i.e. PEC is considered. The tandem configuration ensures the best overall performance due to the highest heat transfer coefficient, which effectively lowers the motor temperature. Moreover, this design guarantees a moderate pressure drop that minimizes the energy required for pumping the coolant, thereby reducing operational costs.
The analysis is performed on a simplified model, focusing on three domains (i.e. windings, stator and fluid). In particular, a perfect contact between the stator and the coils is assumed and a turbulent flow is considered. These lead to a high heat transfer coefficient and enhanced thermal exchange efficiency. These assumptions contributed to the observed low temperature rise in the stator. A future analysis would be conducted to account for the insulating materials used in the construction of the stator as well as more representative conditions such as non-adiabatic conditions and reconsidering the coolant flow rate the values to achieve the desired cooling performance.

ACKNOWLEDGMENTS

This research is supported by A*STAR under its RIE2025 Manufacturing, Trade, and Connectivity (MTC) Industry Alignment Fund-Pre-Positioning (IAF-PP) (Award M22K4a0044).

REFERENCES

[1] R. Saltarelli, L. M. Alves, M. Fasano, and R. M. Afonso, “Joining by forming technology for thermal applications: A case study of finned tube heat exchanger,” *Case Studies in Thermal Engineering*, vol. 59, p. 104551, 2024.

[2] A. Tikadar, D. Johnston, N. Kumar, Y. Joshi, and S. Kumar, “Comparison of electro-thermal performance of advanced cooling techniques for electric vehicle motors,” *Applied Thermal Engineering*, vol. 183, p. 116182, 2021.

[3] X. Wang, B. Li, D. Gerada, K. Huang, I. Stone, S. Worrall, and Y. Yan, “A critical review on thermal management technologies for motors in electric cars,” *Applied Thermal Engineering*, vol. 201, p. 117758, 2022.

[4] J. Shazly, S. Wahsh, and A. Yassin, “Thermal modeling of an afpmsm: A review,” *Journal of Electrical Systems and Information Technology*, vol. 2, no. 1, pp. 18–26, 2015.

[5] H. Wang, C. Zhang, L. Guo, and X. Li, “Novel revolving heat pipe cooling structure of permanent magnet synchronous motor for electric vehicle,” *Applied Thermal Engineering*, vol. 236, p. 121641, 2024.

[6] W. Liu, Y. Dai, J. Zhao, and X. Wang, “Thermal analysis and cooling structure design of axial flux permanent magnet synchronous motor for electrical vehicle,” in *2019 22nd International Conference on Electrical Machines and Systems (ICEMS)*, pp. 1–6, 2019.

[7] J. Zhao, Q. Han, Y. Dai, and M. Hua, “Study on the electromagnetic design and analysis of axial flux permanent magnet synchronous motors for electric vehicles,” *Energies*, vol. 12, no. 18, 2019.

[8] C. Jenkins, S. Jones-Jackson, I. Zaher, G. Pietrini, R. Rodriguez, J. Cotton, and A. Emadi, “Innovations in axial flux permanent magnet motor thermal management for high power density applications,” *IEEE Transactions on Transportation Electrification*, vol. 9, no. 3, pp. 4380–4405, 2023.

[9] J. Tu, G.-H. Yeoh, and C. Liu, “Chapter 7 - practical guidelines for cfd simulation and analysis,” in *Computational Fluid Dynamics (Third Edition)* (J. Tu, G.-H. Yeoh, and C. Liu, eds.), pp. 255–290, Butterworth-Heinemann, third edition ed., 2018.

[10] A. F. Al-Neama, N. Kapur, J. Summers, and H. M. Thompson, “An experimental and numerical investigation of the use of liquid flow in serpentine microchannels for microelectronics cooling,” *Applied Thermal Engineering*, vol. 116, pp. 709–723, 2017.