Numerical study of the impact of different cooling arrangements on a high length/diameter ratio motor for electric commercial vehicle

Salvatore La Rocca
University of Nottingham
Faculty of Engineering
Nottingham, United Kingdom
salvatore.larocca@nottingham.ac.uk

Antonino La Rocca
University of Nottingham
Faculty of Engineering
Nottingham, United Kingdom
a.larocca@nottingham.ac.uk

Adam Walker
University of Nottingham
Faculty of Engineering
Nottingham, United Kingdom
Adam.Walker@nottingham.ac.uk

Gaurang Vakil
University of Nottingham
Faculty of Engineering
Nottingham, United Kingdom
Gaurang.Vakil@nottingham.ac.uk

David Gerada
University of Nottingham
Faculty of Engineering
Nottingham, United Kingdom
David.Gerada@nottingham.ac.uk

Mahir Al-Ani
University of Nottingham
Faculty of Engineering
Nottingham, United Kingdom
mahir.al-ani@nottingham.ac.uk

Tianjie Zou
University of Nottingham
Faculty of Engineering
Nottingham, United Kingdom
Tianjie.Zou@nottingham.ac.uk

Ramkumar Ramanathan
University of Nottingham
Faculty of Engineering
Nottingham, United Kingdom
Ramkumar.Ramanathan@nottingham.ac.uk

Chris Gerada
University Of Nottingham
Faculty of Engineering
Nottingham, United Kingdom
Christopher.Gerada@nottingham.ac.uk

Alastair McQueen
Cummins Ltd
Cummins Corporate R&T
Peterborough, United Kingdom
alastair.mcqueen@cummins.com

Krzysztof Paciura
Cummins Ltd
Cummins Corporate R&T
Peterborough, United Kingdom
krzysztof.paciura@cummins.com

***Abstract*—Computational Fluid Dynamics (CFD) was used to carry out a fluid flow and thermal investigation of a high power dense traction machine for electric commercial vehicle. One of the features of the machine design considered is the high Length/Diameter (L/D) ratio; this can represent a further challenge when performing the thermal design; indeed the heat dissipation from the stator core can represent an issue. The aim of this work is to identify an effective cooling configuration capable of enhancing the cooling of the stator and maintaining the operating temperature of the whole machine within the allowable limits. In this paper CFD analyses were therefore used to assess the thermal capability of several cooling arrangements; numerical results of each case are presented and compared. In particular an alternative cooling option, consisting in the implementation of heat pipes between the machine's stator teeth, was considered. Due to their simple structure and high heat transfer coefficients, heat pipes are not invasive but at the same time can dissipate large amount of heat. CFD results show how a significant temperature drop of the machine's core can be achieved.**

Keywords—Thermal, Computational Fluid Dynamics Cooling, High power density, Heat transfer

I. Introduction

The regulations implemented by the European commission impose the reduction of the emissions by the 60% by the 2050 [1]. Currently the impact of the road vehicles (both light and heavy duty) on the emission of CO_2 represent up to the 40% of the total emissions within the EU [2]. Indeed, it is expected that a reduction of the conventionally fuelled cars by 50% is required by the 2030 [3].

Electrically driven vehicles are expected to be playing the major role for achieving those targets. The automotive industry is therefore moving towards the implementation of high power traction machines capable of replacing the current internal combustion engines [3]. The machine designs developed for such scope often result to be very high power dense; that means that, despite the high efficiency achievable (typically over 95%), the heat rejection can still be significant and can reach peaks in the order of tens of kW.

One of the main challenges of the design process of such machines is making sure that the operating temperatures would not exceeds the limits; this has a direct impact on the life time of the components, the efficiency and the reliability of the whole system. The lack of space available on the vehicle for the motor can further reduce the heat transfer surface available, making the heat dissipation even more challenging. More aggressive cooling configurations, particularly based on direct liquid cooling methods are then required. [4] – [7]

This work will describe the thermal analyses carried out on a high power dense traction machine for a commercial vehicle under different cooling scenarios. These aim to identify the optimal cooling configuration for the given design.

The numerical CFD study was performed using a commercial software package (Ansys Fluent) and the models developed included both solid and fluid domains in order to capture the interaction between them.

II. CFD Methodology

Numerical analyses were carried out by means of CFD techniques to investigate the thermal performance of the existing machine cooling design; this consisted of a liquid cooling design which employs a spiral water jacket with rectangular ducts located within the external housing. The initial analysis of the existing cooling jacket design allowed identifying the most critical areas from a thermal point of view, based on the temperature distribution obtained. This stage was important to assess potential cooling improvements to be considered in order to improve the thermal performance of the machine under investigation.

Based on the machine design, the fluid domain of the water jacket was generated within Ansys Design Modeler as show in Fig. 1 below; Table I reports the inlet conditions of flow rate and temperature of the water jacket.

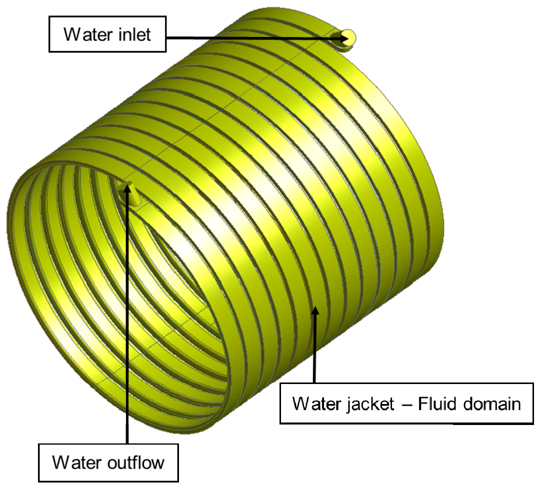

Fig. 1 - water jacket fluid domain

Table I - water jacket, inlet conditions

Inlet conditions	
Volume flow rate	Temperature
6 L/min	85 °C

The aim of this analysis was the determination of the pressure drop and the convective heat transfer coefficient within the jacket; this was then implemented in the full machine conjugate model as thermal boundary condition at the housing walls in order to reduce the overall computational cost and time of the machine thermal model. The water jacket fluid domain was discretised with Ansys Mesher; since the geometry had a regular cross section, the sweep meshing method and hexahedral cells were adopted in order to achieve a high numerical accuracy and to reduce the number of elements. The final mesh counted 860,000 elements with a simulation time of about one hour; Table II below summaries the boundary conditions employed and the results achieved (heat transfer coefficient, HTC, and pressure drop) with Ansys Fluent version 17.2 [8]

Table II - water jacket, boundary conditions and results

Boundary condition	**Results**	
Heat flux – jacket walls	HTC	Pressure drop
1000 W/m^2	1400 W/m^{2}K	0.07 – 0.1 bar

Once the fluid domain of the water jacket was thermally investigated and the HTC calculated, the conjugate model of the machine under investigation was developed taking into account all the internal fluid and solid domains as shown in Fig. 2 below.

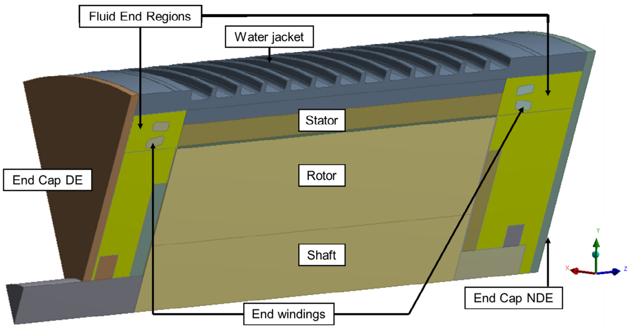

Fig. 2 - CFD conjugate model, fluid and solid domains, 45° periodic sector

As can be seen, a 36° sector only was considered and periodic boundary conditions were employed in order to reduce the computational cost, hence calculation time of the CFD model. While components such as bearings and stator windings were modelled as bulk solids (with overall thermal properties assigned), a detailed end windings representation was developed with dimensions and topology comparable with the actual machine end windings as reported in Fig. 3; this was done in order to achieve more accurate and detailed predictions of the complex fluid flow and thermal phenomena occurring in the end regions of the machine.

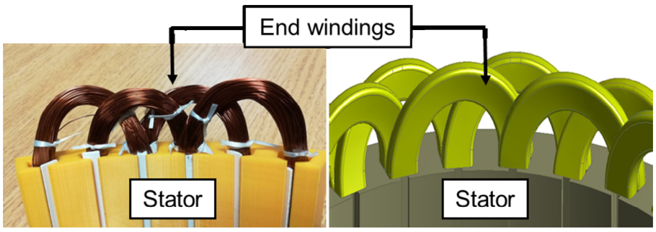

Fig. 3 - end windings geometry modelling

The fluid domains were split into stationary and moving domain (which contained all the rotating parts such as shaft and rotor) in order to implement the rotation using the Multiple Reference Frame (MRF) approach and perform a steady-state analysis. In the case under investigation the rotational speed was set at 2,800 rpm and the realizable k-Ɛ turbulence model was implemented since it is more suitable for rotation problems and it can generally provide better performance than the standard k-Ɛ model [8].

Thermal boundary conditions allowed the modelling of heat generation due to electromagnetic losses occurring within the machine; these were implemented within the CFD model as energy source in the solid cell zones (W/m^3) with a uniform volumetric heat generation rate. Fig. 4 shows the loss distribution within the machine, as it can be seen 96% of the total power loss occurs in the stationary components. As previously discussed, at the water jacket walls a thermal boundary condition of 1,400 W/m^{2}K was used; it was assumed that the heat was dissipated through the cooling jacket only.

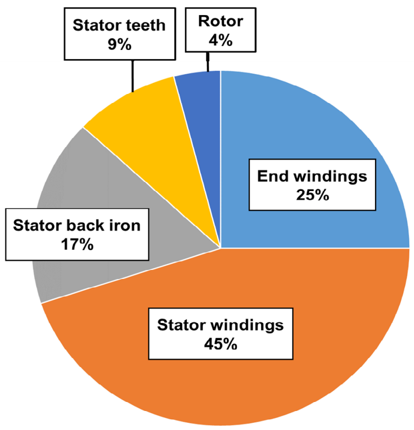

Fig. 4 - power loss distribution

Several meshing techniques were used simultaneously to achieve a high quality and computationally efficient meshes. Hexahedral cells were adopted for simple geometry solid regions (stator, rotor, shaft and bearings) in order to achieve a high numerical accuracy and to reduce the number of elements. More complex domains (such as the end regions and housing) were discretised using tetrahedral elements due to their capability of better discretising curved geometries. A total of 6.2 million of elements were used to discretise the 36° periodic sector and the numerical converged solution was achieved in 12 hours.

III. CFD RESULTS

Results of temperature distribution and magnitude were used to assess the effectiveness of the cooling configurations; this also provided an insight into the maximum operating temperature and the location of hot spots within the electrical machine. Fig. 5 and Fig. 6 below report the contours of temperature of the solid regions for the case under investigation. As it can be noted, results of temperature distribution were in accordance with the location and magnitude of the electrical losses: due to higher copper losses for instance, higher temperatures were found in the stator core and end windings.

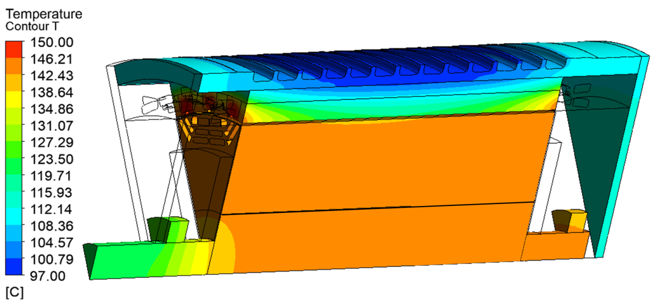

Fig. 5 - CFD conjugate model, machine contours of temperature

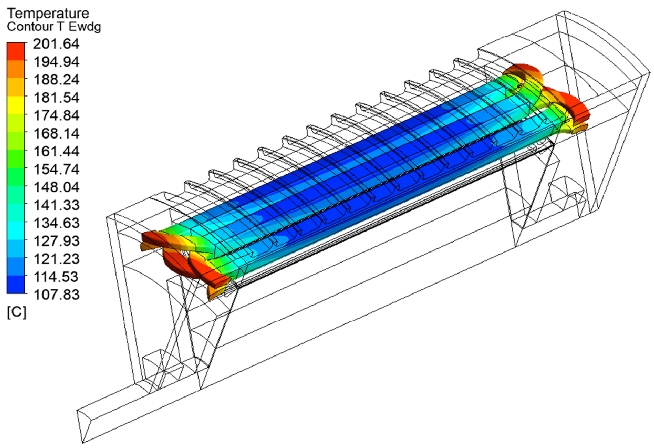

Fig. 6 - CFD conjugate model, machine contours of temperature, windings

Numerical results also highlighted the cooling effectiveness of the water jacket along the axial direction. However, the temperature of the end windings was predicted to be above 200°C (Fig. 6); numerical results indicated the end windings as potential critical area for the thermal point of view under the current cooling conditions. Further investigations of direct oil cooling arrangements for the end windings were therefore considered to be used simultaneously with the existing water jacket; the developed CFD model was also used to investigate and assess the cooling capability of alternative arrangements of the stator. The aim of this work was to consider and find new practical cooling solutions for electrical machines with a high L/D ratio (2 for the case analysed) where the cooling of the machine middle section can be very critical and challenging in case a water jacket system is not used. In previous works [9] where the L/D machine ratio was about 0.5, the only oil flooding of the end region was enough to dissipate effectively the heat generated inside the stator.

Several cooling options, with and without the existing water jacket, were took into account and investigated as summarised in Fig. 7 below. A practical and immediate solution to the high end windings temperatures issue highlighted in Fig. 6 was the addition of direct oil cooling in the end region to the existing water jacket design (Case 1). Then, several CFD analyses were carried out to investigate the impact of the water jacket removal on the thermal performance of the machine: this was considered in Case 2, where the only end region oil cooling was used. In Case 3, in combination with the oil flooding of the end region, an alternative axial cooling arrangement was used; this consisted of heat pipes located in the stator core.

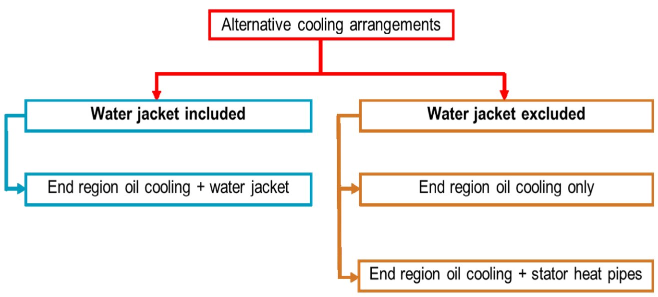

Fig. 7 - alternative cooling arrangements considered

A. Case 1: End region oil cooling and water jacket

Similarly to the analysis previously presented, the water jacket cooling was considered with a thermal boundary condition of 1,400 W/m^{2}K used at the housing/jacket walls. As depicted in Fig. 8 below, the direct oil cooling of the end windings was considered by assuming and applying a conservative convective heat transfer coefficient of 1,500 W/m^{2}K [10] – [12] at the end windings walls.

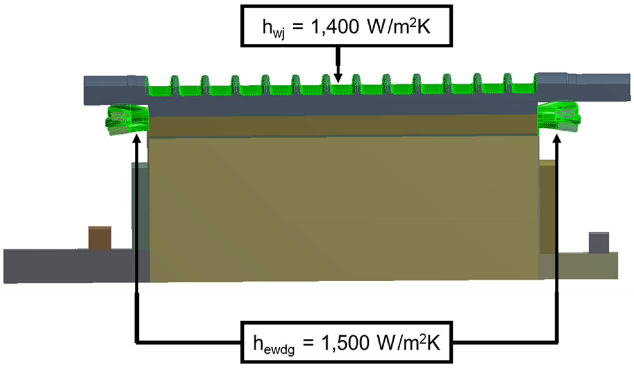

Fig. 7 - Case 1, convective heat transfer boundary conditions

Numerical results of temperature distribution across the machine (Fig. 9) show how the combination of direct oil cooling over the end windings and water jacket is able to significantly reduce the operating machine temperature below 128°C.

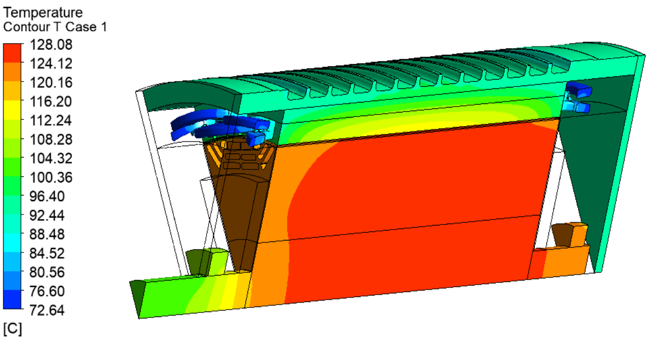

Fig. 9 - CFD conjugate model, machine contours of temperature, Case 1

B. Case 2: End region oil cooling only

This case was considered to better highlight the challenge and the requirement of axial cooling when designing an electrical machine with a high L/D as in the case under investigation. It was assumed that the heat was entirely dissipated only through the direct oil cooling over the end winding surfaces; no cooling along the axial direction was therefore considered. The same thermal boundary condition to model the oil cooling of Case 1 was used while the water jacket walls were set as adiabatic. Results of temperature are shown in Fig. 10 below where it can be seen how the only end region cooling is not able to provide any effective cooling to the middle section of the machine without considering any additional axial cooling options.

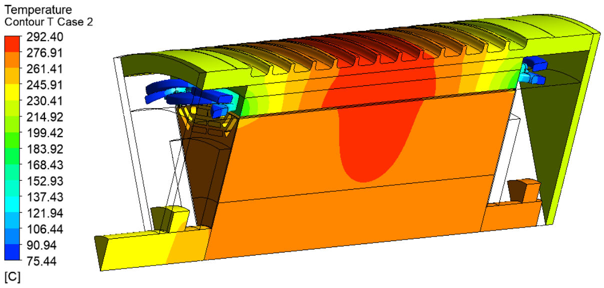

Fig. 10 - CFD conjugate model, machine contours of temperature, Case 2

C. Case 3: End region oil cooling and stator heat pipes

In the case under investigation, the heat pipes were modelled as bulk solids (2 mm radius) with an equivalent thermal conductivity of 8,000 W/mK [13]; they were located in-between the stator teeth (Fig. 11) in low flux density regions in order to minimise the impact on the electromagnetic performance. The aim of the investigation was to establish high conductivity paths in order to push heat away from the middle section towards the oil flooded end region.

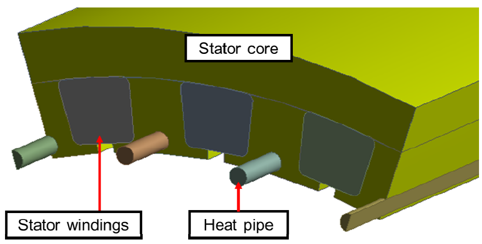

Fig. 11 - Case 4, heat pipes location in the stator core

Heat pipe is a heat transfer device based on the principle of thermal conductivity and phase transition; Fig. 12 below reports a schematic view of a generic heat pipe and its working principle. A heat pipe is a sealed pipe made with copper or aluminium and it is partially filled with a working fluid (water or ethanol for example) which is chose in function of the expected temperature range. The working fluid evaporates absorbing thermal energy from the hot side (1 - stator core); due to the pressure difference between the hot and cold side (end region oil flooded), the vapour migrates along the cavity towards the cold side (2) and condenses by transferring heat to the wick (3). The working fluid is then flowing back by capillarity through the wick to the hot side end (4) to start over again the cycle. Due to the high heat transfer coefficients for the boiling and condensation processes, heat pipes are highly effective thermal conductors and can approach very high thermal conductivity values.

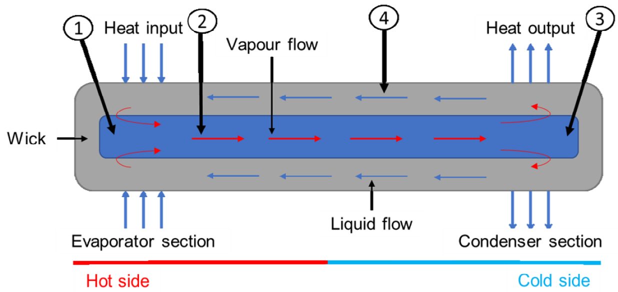

Fig. 12 - heat pipes schematic

Numerical results of temperature distribution and magnitude were computed in order to assess the effectiveness of the proposed cooling design; this is reported in Fig. 13 below. In comparison to the results of Case 2 (Fig. 10), where no axial cooling was used, it can be seen that a significant temperature drop of the middle section of the machine (about 40°C) could be achieved.

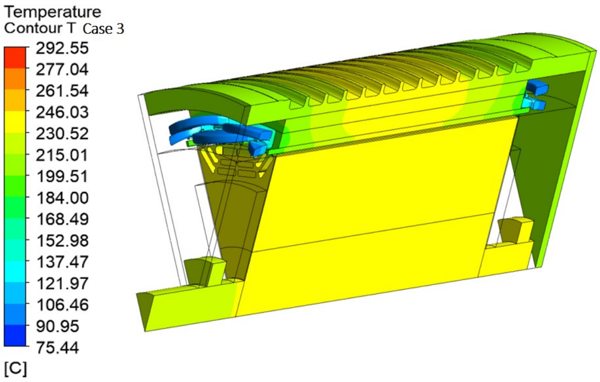

Fig. 13 - CFD conjugate model, machine contours of temperature, Case 3

IV. Conclusion

The work presented in this paper describes a CFD investigation of the thermal performance of a high power dense traction machine for electric commercial vehicle applications.

Numerical results of temperature distribution and magnitude computed from the comparison between the several cooling options show how Case 1 (combination of end region oil cooling and water jacket) was the most effective cooling arrangement for both the end regions and the middle section of the machine; the maximum temperature achieved was about 128°C in the rotor core and magnets and overall temperatures were below the maximum insulation temperature allowed.

Results also highlight the challenge and requirement of axial cooling of electrical machines with high L/D ratio. In the case under investigation if no axial cooling was used (Case 2, end region oil cooling only) very high temperatures (292°C) were predicted in the middle section.

Based on that, alternative axial cooling options to the existing water jacket design were considered and investigated. Heat pipes (Case 3, combination of end region oil cooling and heat pipes) were modelled and introduced between the machine's stator teeth. Heat pipes have the advantage to contain no mechanical moving parts and typically require no maintenance. Mostly important they are highly effective thermal conductors due to the high heat transfer coefficients for the boiling and condensation processes; in this work the aim was to provide high conductivity thermal paths from the machine's core to the oil flooded end regions.

CFD analyses were used to assess the cooling capability of the heat pipes for an electrical machine with high L/D ratio. Numerical results show how a significant temperature drop of the machine's core can be achieved.

If no other axial traditional cooling arrangements (such as water jacket) are available, the use of heat pipes in electrical machines with high L/D ratio could be considered as a promising cooling option which needs further investigation in terms of heat pipe model and optimization of its location within the electrical machine; this will be object of future work.

Acknowledgment

The authors gratefully acknowledge Cummins Generator Technologies for the support on this work.

References

[1] European Commission, Strategy for reducing Heavy-Duty Vehicles' fuel consumption and CO2 emissions, COM 2014 (285)

[2] European Commission, A European Strategy for Low-Emission Mobility, COM (2016), 501.

[3] EU Transport GHG: Routes to 2050 II 15 European Commission, White Paper: Roadmap for a single European transport area. COM (2011) 144

[4] P. Arumugam, Z. Xu, A. La Rocca, G. Vakil, M. Dickinson, E. Amankwah, T. Hamity, S. Bozhko, "High-speed Solid Rotor Permanent Magnet Machines: Concept and Design", IEEE Trans on Trans. Elec, Vol. 2, pp 391-399 Sept 2016.

[5] X. Jannot, J. Vannier, C. Marchand, M. Gabsi, J. Saint-Michel, D. Sadarnac, "Multiphysic Modelling of a High-Speed Interior Permanent-Magnet Synchronous Machine for a Multiobjective Optimal Design, IEEE Trans. on Energy Conv, Vol. 26, pp 457-467, June 2011

[6] B. Dong, K. Wang, H. Bancheng, S, Zheng. "High-speed permanent magnet motor with magnetic bearings: Multi-physics analysis, cooling deisgn and experiment" 19th International Conference on Electrical Machines and Systems (ICEMS), 13-16 Nov 2016

[7] Z. Kolondzovski, A. Belahcen, A. Arkkio, "Multiphysics design of a high-speed permanent-magnet machine, Applied Thermal Engineering, Vol 29, Issue 13, pp 2693-2700, Sept 2009

[8] Fluent, "ANSYS FLUENT User's Guide v17.2", 2016

[9] S.Bozhko, T. Yang,J.M. Le Peuvedic, P. Arumugam, M. Degano, A. La Rocca, Z. Xu, M. Rashed, W. Fernando, C. Hill, C. Eastwick, S. Pickering, C. Gerada, P. Wheeler. *Development of Aircraft Electric Starter-Generator System Based-On Active Rectification Technology* IEEE transactions on transportation electrification. 201

[10] A. La Rocca. *Thermal analysis of a high speed electrical machine.* PhD thesis, University of Nottingham, 2016.

[11] L. Ye, F. Tao, L. Qi and W. Xuhui, "Experimental investigation on heat transfer of directly-oil-cooled permanent magnet motor," *2016 19th International Conference on Electrical Machines and Systems (ICEMS)*, Chiba, 2016, pp. 1-4.

[12] Polikarpova, M., Liquid Cooling Solutions for Rotating Permanent MagnetSynchronous Machines, 2014, Lappeenranta University of Technology: Finland.

[13] D. Worwood, Q. Kellner, M. Wojtala, W.D. Widanage, R. MGlen, D. Greenwood, J. Marco. A new approach to the internal thermal management of cylindrical battery cells for automotive applications, Journal of Power Sources, 2017