Numerical investigations of convective phenomena of oil impingement on end-windings

Peter H. Connor[1] ✉, Antonino La Rocca[1], Zeyuan Xu[1], Carol N. Eastwick[1], Stephen J. Pickering[2], Chris Gerada[3]

[1]*Fluids and Thermal Engineering Research Group, Faculty of Engineering, University of Nottingham, UK*
[2]*Composites Research Group, Faculty of Engineering, University of Nottingham, UK*
[3]*Power Electronics, Machines and Control Research Group, Faculty of Engineering, University of Nottingham, UK*
✉ *E-mail: peter.connor@nottingham.ac.uk*

Abstract: A novel experimental rig for analysing intensive liquid cooling of highly power-dense electrical machine components has been developed. Coupled fluid flow and heat transfer have been modelled, using computational fluid dynamics (CFD), to inform the design of a purpose-built enclosure for optimising the design of submerged oil jet cooling approaches for electrical machine stators. The detailed modelling methodology presented in this work demonstrates the value in utilising CFD as a design tool for oil-cooled electrical machines. The predicted performance of the final test enclosure design is presented, as well as examples of the sensitivity studies which helped to develop the design. The sensitivity of jet flow on resulting heat transfer coefficients has been calculated, while ensuring parasitic pressure losses are minimised. The CFD modelling will be retrospectively validated using experimental measurements from the test enclosure.

1 Introduction

The current increasing trend in demand for high power density electrical machines, especially from the aerospace industry [1], has required machines to be significantly reduced in size and weight. The thermal management demand on these machines warrants analyses with greater detail and accuracy on increasingly more novel and complex thermal design concepts. An accurate prediction of a machine's thermal behaviour can considerably speed up the design process and lead to more reliable and efficient systems.

The major challenge in the thermal design of high power dense machines is that large amounts of heat need to be dissipated from very small surface areas. Direct liquid cooling arrangements, such as submerged oil-jet cooling, enable high heat dissipation due to high convective heat transfer coefficients. This enables higher current densities to be utilised [2–9]. However, the thermal gains must be balanced against the increased mass and design complexity an intensive cooling system brings, when compared to more conventional electrical machine cooling arrangements.

There is a lack of robust research regarding the performance of oil cooling in the complex end-winding regions that is required to design an optimal system. The modelling of the complex fluid flows and detailed novel geometries, such as jet and end-winding interactions, is not well suited to lumped parameter thermal network analyses, due to their lack of fidelity and availability of appropriate correlations. To assess the benefits vs. cost of using liquid oil-jet cooling, coupled heat transfer and fluid analyses via direct experimental measurement or computational fluid dynamics (CFD) are the most suitable tools available. Other submerged oil delivery methods, other than individual jets exist, such as slots or plug flow entry through large openings. While these could be investigated using the same experimental setup and modelling methodologies, presented within this paper, this case study focusses on jets.

To assess a novel approach for cooling a stator using oil-jet impingement, an experimental test enclosure has been developed. This will allow the investigation of fluid flows and their impact on localised heat removal from a representative stator section. A test section for motorettes is more flexible and cheaper than building prototype entire machines. The test enclosure has involved extensive planning, by an inter-disciplinary team. This ensures a robust method for testing the cooling performance in a way that is representative of realistic electrical winding design with appropriate manufacturing methods, materials selection and mechanical structure. CFD has been used to ensure the design meets the complex fluid and thermal cooling requirements. The modelling has also helped to identify the key variable parameters which influence oil-jet cooling of a stator. This ensures that the test enclosure is designed to allow for both major and minor adjustments of the principal parameters. This approach facilitates a coupled experimental and CFD approach for the design and optimisation of oil-jet cooling concepts [10].

2 Methodology

CFD is better suited at modelling complex jet–fluid interactions with irregular winding surfaces than the alternative analytical modelling methodologies. However, due to this complex interaction, there may be uncertainties over the numerical findings. An experimental validation is therefore required to prove the consistency of the numerical predictions carried out.

For the reasons above, a test enclosure was developed to experimentally investigate the behaviour of the submerged oil jets in conjunction with a representative CFD model; indeed, the computational domain replicates the geometry of the test enclosure. The enclosure consists of a box containing a 90° sector of a wound stator. At each end of the enclosure multiple inlets supply oil to plenums, on which the jets are mounted. The oil can then flow through the stator core leaving the enclosure from the central plane of the stator. An external view of the manufactured test enclosure is shown in Fig. 1.

Some details such as plenum design and inlets sizing were carried out with the aid of the CFD in order to guarantee a uniform distribution of the flow among all the jets and to minimise the pressure drops across the system. The key advantages of using the test enclosure are the following:

- *Flexibility:* jets positions, type and layout can easily be changed; also multiple stator and windings configurations can be tested.
- *Cost:* partly due to the flexibility of its design, the test enclosure allows a significant reduction of costs, both in terms of time and

money, compared to the manufacture of a full machine prototype.

The physical experimental test enclosure will go on to both provide validation data for the CFD modelling presented here (and wider future CFD modelling), as well as to produce its own detailed fluid and thermal experimental data, which will be presented in future work.

In the following sections, the parametrised computational model created will be described and finally the numerical findings will be discussed.

3 Numerical modelling

The CFD model is a fluid-only domain which has been generated to help to inform the design of the test enclosure. This model includes simplified toroidal shaped bulk-end-windings. Similar, simplified end-winding geometries have been used in numerous thermal models [11–13]. In the next section, the detailed CFD modelling setup for oil-jet impingement on electrical machine end windings, including meshing approaches, setting of fluid parameters, turbulence modelling, design considerations and methods for post-processing, relevant to electrical machines, is presented.

3.1 Model overview – geometry and meshing

A CFD model has been created to represent the 90° stator sector for the experimental enclosure. Due to the relatively high modelling cost of CFD, simplifications to the geometry are made to reduce overheads. User expertise must determine the extent of simplifications, while maintaining the quality of the model for an accurate solution. In this case, the model size is reduced, due to the symmetry which appears in the geometry and will be exhibited in the flow. Symmetry planes are used to simplify the model down to a 45° stator sector and half of the axial core. This reduces cost, while appropriately representing the full test enclosure geometry (see Fig. 2). Small geometrical features, such as machining radii or bolt heads are removed from the CFD domain.

A range of investigations have been made using the models presented in this paper. These were aided through the consideration of the intended parametric changes before the geometry and meshing stages. The geometry was built so the pipe diameters and plenum sizes were variable. Similarly, the mesh was constructed to enable robust and quality meshing to adapt to the aforementioned geometrical changes [10].

A hybrid mesh has been generated with a combination of hexahedral cells, within the pipework and jet cores, and tetrahedral cells which fill the remainder of the fluid domains. For narrow jets and a fluid with the viscosity of oil, a considerable pressure drop is experienced within the jet itself and up/downstream of the jet at the sudden contraction and expansion regions, respectively. To accurately resolve the complex flows in these regions, the mesh is fine in these regions. Approximately 10 cells are modelled across the diameter of the jet.

The meshes for the cases presented here have ∼10 million cells. Several models were run with a combination of geometrical and fluid parametric changes. Geometric changes result in changeable mesh counts, but the model is built to maintain the quality of meshing.

3.2 CFD solver setup

The model is solved within the finite volume software ANSYS Fluent, version 17.2. The fluid-only model is solved using the standard k–ε turbulence model, with the enhanced wall function enabled. This is suitable for enclosed flows, such as the one found in this work. The energy equation was enabled for heat transfer.

The test rig is to be operated for testing a range of liquid coolants. The work presented in this paper focuses on a single oil, for which the physical fluid and thermal characteristics are known. Within the solver, a fluid is created to represent the intended test oil, to ensure accurate flow distributions, pressure drops and temperature distributions are predicted. As the test enclosure being
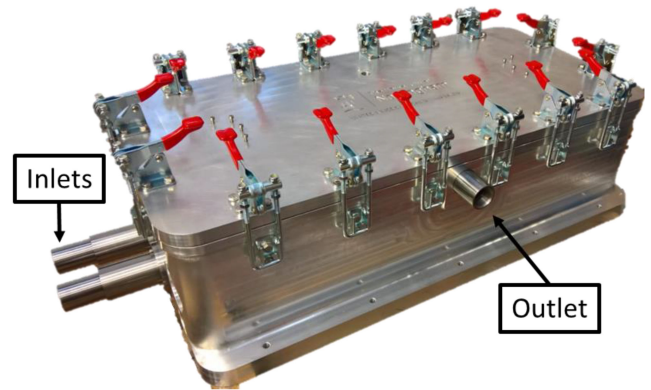

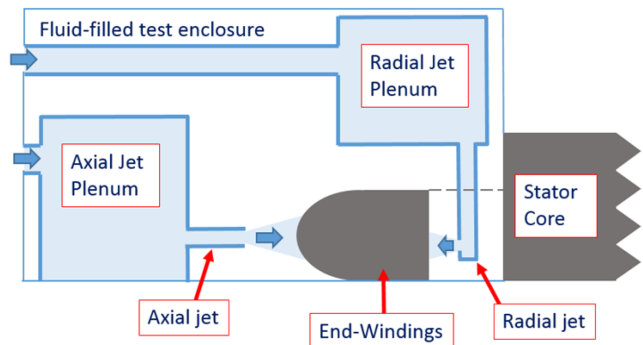**Fig. 1** *Manufactured test enclosure*

Fig. 2 *2D schematic of jet and end-winding region*

designed is to be fully flooded, all cell zone volumes within the model are specified with the oil in this fluid-only model.

Mass flow inlets are specified at the oil pipe inlet surfaces and are set with a prescribed flow rate. The inlet temperature of the oil is specified. These are later varied as a parametrically variable feature. The outlet of the model is created as a pressure outlet. The outlet pressure at this surface is set to ambient pressure. Symmetry boundary conditions are used on the two side boundaries, about which the geometry is mirrored. The source for heat transfer in this fluid-only model is the stator end-winding wall surface. This is set at a constant temperature. This enables heat transfer coefficients to be computed on the end-winding surfaces.

The main output parameters monitored from the CFD modelling are the pressure distributions within the system, mass flow rate distributions between the jets and surface heat transfer coefficients on the end windings are computed. Results are presented on a range of images and plots in the following section.

The PC used for this investigation has dual core Intel Xeon processors, with 24 cores combined. A single design point for this model runs in ∼5 h.

4 Numerical results

The results in this paper have been divided into two sections. Firstly, the final experimental test section and jet fluid and thermal design are presented and discussed. Secondly, the flexibility of CFD modelling and its use for understanding the complex system behaviour is explored and sensitivities to selected geometric and fluid flow changes are presented.

All values are non-dimensionalised based on the maximum calculated value for a given variable.

4.1 Results from CFD modelling of final test enclosure topology, fluid flow and heat transfer.

In this section, CFD modelling results from the final test enclosure, plenum and jet design are presented. Fig. 3 shows a 2D contour plot of velocity taken through the fluid domain (image matches schematic shown in Fig. 2). It shows that the highest velocities are within the jets. The flow dispersion and impingement on the end-winding surfaces is visible from both the radial and axial jets. The
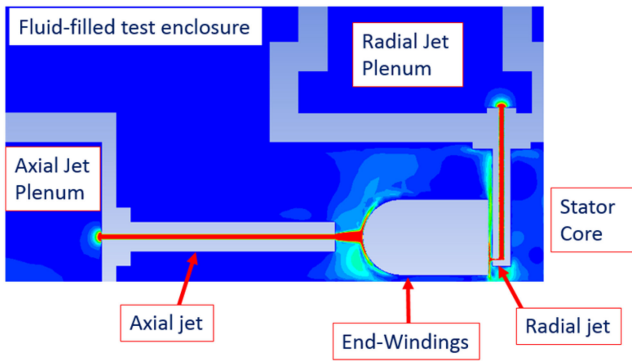
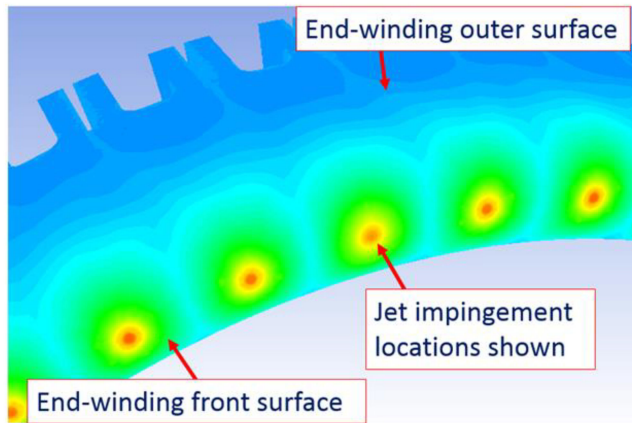**Fig. 3** *2D Contours of Velocity in end-winding region*

Fig. 4 *Distribution of heat transfer coefficients shown on outer surfaces of end winding*
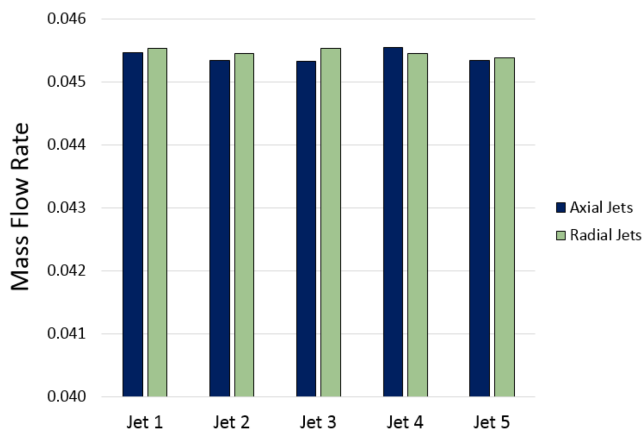

Fig. 5 *Non-dimensionalised flow distribution between jets from both axial and radial plenums*

Table 1 Non-dimensionalised pressure distribution through system

Location	Peak pressure
radial plenum pipe inlet	1.00
axial plenum pipe inlet	0.69
radial plenum	1.00
axial plenum	0.69
test box	0.25

consistent reference temperature for all cases can better highlight the variation of the heat transfer coefficients, for comparison purposes

$$h_{surf} = \frac{Q}{A}(T_{wall} - T_{inlet}) \qquad (1)$$

where: h_{surf} is the surface heat transfer coefficient (W/m^2 K); Q is the heat loss through the surface (W); A is the surface area (m^2); T_{wall} is the average wall temperature (K); T_{inlet} is the temperature of the oil at system inlet (K)

The plenums and jets have been designed to ensure delivery of a uniform flow distribution between the jets. This is shown by Fig. 5. There is <0.5% change between the minimum and maximum jet mass flow rates for both the axial plenum jet group and the radial plenum jet group. This demonstrates the appropriate sizing of the plenums and oil delivery pipework. This is further explored next.

The distribution of pressures in any experimental fluid setup is important. In this case, the design requirements stipulate a maximum operating pressure. Careful design ensured good heat transfer was achieved, through the use of jets, without the peak pressure exceeding required limits. The distribution of pressure within the system is shown in Table 1. The highest pressure drop occurs across the jets, between the plenums and test enclosure. This large pressure drop helps to balance the flow between the jets, as seen in Fig. 5.

It must also be noted that designing pressurised test enclosures or machines can be subject to specific pressure vessel health and safety regulations. In this case, while the operating pressure is relatively low, the experimental rig has a large surface area on the lid. For this reason, the pressures are closely monitored for the inner plenums and the main box. The data produced by the CFD is subsequently used for structural finite element analyses for the rig design.

4.2 Parametric use of CFD model during test enclosure design and analyses

The CFD model presented in this paper was built with the flexibility to enable some parametric changes to be investigated. In this section, geometric changes to the delivery pipe diameters and plenum sizes, as well as varying the system oil flow rates, are explored to demonstrate the value of CFD modelling for sensitivity analyses. All cases in this section retain the same stator geometry, overall test enclosure size, constant jet sizing, spacing and proximity to the end windings.

Firstly, the diameter of the oil delivery pipes to the plenums were investigated (see Fig. 1). The diameter of these pipes is important as they contribute to the system pressure, which is limited. The flow of fluid through the pipes causes internal fluid friction which exhibits itself as a pressure loss along the pipe. The expansion of this fluid into the plenums also has a pressure drop associated with it. Both of these losses are related to the fluid velocity. Increases in the pipe diameters reduces the fluid velocity, at constant mass flow rate, which can help to limit system pressures. In this pipe diameter sensitivity study, the operating flow rate from Section 4.1 is maintained constant. The delivery pipe pressures are reduced from the full diameter down to a half and quarter diameter. The maximum system pressure is plotted against changing pipe diameter in Fig. 6. The squared relationship between fluid velocity and pressure drop causes a significant rise in system

fluid momentum disperses very quickly, due to the proximity of the jet to the end winding, as well as the relatively high viscosity of the oil. The post-impact turbulent fluid around the end winding helps to raise the heat transfer coefficient outside the initial impact zone (see also Fig. 4). It is this behaviour that is particularly difficult to predict analytically, due to the highly geometry dependent fluid behaviour, which is why CFD modelling was used for this case.

The CFD modelling has been used to simultaneously solve the fluid flow behaviour and the heat transfer from the winding surface. Fig. 4 shows 3D contours of surface heat transfer coefficient plotted onto the end-winding surfaces. The front face of the end winding, facing the axial jet, is of interest. The jet diameter and proximity are chosen to raise the regions of high heat transfer across much of the end-winding front surface. Similarly, the circumferential spacing of the jets shows a good spread of fluid cooling around the winding. Surface heat transfer coefficients, presented within this paper, are calculated using the fluid inlet temperature as the reference temperature, see (1). Choosing a
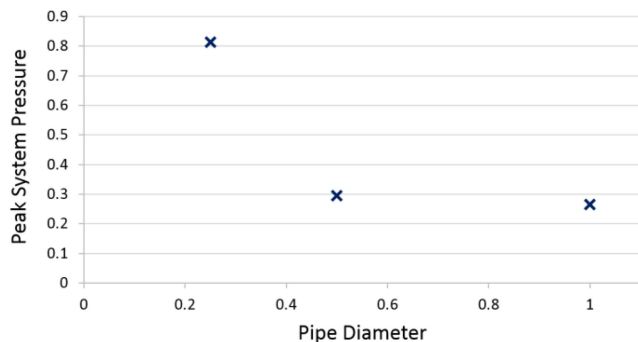
Fig. 6 *Non-dimensionalised change in peak pressure caused by changes in the diameter of the oil delivery pipes to the jet plenums*
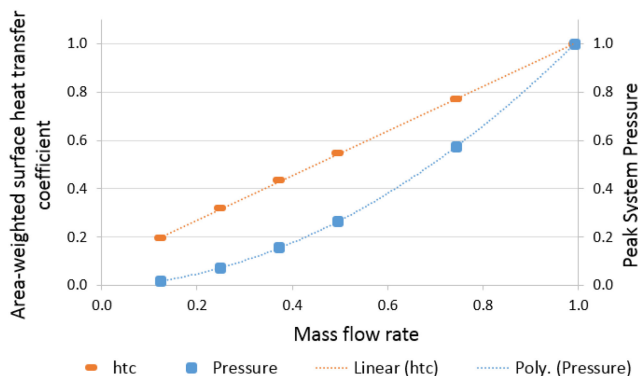

Fig. 7 *Non-dimensionalised graph to show how the peak system pressure and area averaged surface heat transfer coefficient on the stator end-winding surfaces are affected by the mass flow rate*

pressure when the pipe diameter is reduced from half to a quarter of the final design diameter. The design diameter (1.0) is chosen, due to the relative insensitivity to pressure drop at this pipe size, which is clearly seen in Fig. 6.

The second parametric investigation sought to investigate the effect of changing mass flow rate through the system. As the flow rate increases, velocities increase. As seen above, the pressure drops throughout the system increase with velocity as well as the relationship between jet velocity and heat transfer coefficient on the end winding caused by the impinging jet. The flow rate was varied from a quarter to double of the operating flow rate (the operating flow rate is shown as a non-dimensionalised mass flow rate of 0.5 in Fig. 7). The CFD model was run to output two parameters; peak system pressure and area-weighted average surface heat transfer coefficient on the stator end winding surfaces are plotted on a graph in Fig. 7. These two output parameters are the most important aspects in this experimental rig design. The rig must be designed to ensure it can operate to generate jet flows to study surface heat transfer coefficients within the range of interest. However, it is also limited by the maximum pressure the rig may operate at.

The squared relationship between system flow rate and peak pressure is seen in Fig. 7. A limit in the design requirements for the operation of the test enclosure dictates the system maximum flow rate and operating pressure. However, it is important to know what region of the system flow rate vs. pressure curve the system is operating within. The gradient of the pressure curve changes relatively slowly, which enables a wide range of flow rates to be investigated.

The increasing system flow rate increases the velocity of the axial and radial jets, which impinge onto the stator end-winding surfaces. The resulting linear relationship, shown in Fig. 7, provides an estimate of the cooling performance this enclosure design can yield for the given stator topology. It is vital to ensure the enclosure design replicates the intensive cooling performance the rig has been designed to study.

The distribution in mass flow rate between the jets in the axial and radial plenums was confirmed to remain consistent, with negligible difference in flow between jets, for all total system mass flow rates investigated.

The third parametric investigation carried out sought to check the plenums operated correctly. It is important to investigate the entire fluid system, especially where machine design space requirements may dictate narrow regions which could impact the jet distribution. In this case, the plenums are over-sized for practical experimental reasons, but its sizing was investigated using the CFD model. Plenums are typically implemented to allow the incoming fluid to reach a uniform pressure. In this case, it is important to ensure the jet entry pressure is constant, which should lead to a uniform flow distribution between the jets. The CFD model was used to investigate narrower plenum distances, in the axial direction between the axial plenum delivery pipe and the jets (see Fig. 2). The plenum size was incrementally reduced from 100% down to 10% width. It was found that the effect of plenum width on peak plenum pressure and flow distribution from the jets was <1% change between the narrowest and widest plenum modelled. For this reason, the plenum is large enough to not have an impact on jet flow distribution. This is due to the fact the relatively viscous oil causes the submerged oil momentum to be dispersed quickly. The size, number and distribution of oil delivery pipes also aids the performance of the plenum, by having been designed to minimise system pressures by reducing flow velocities of the upstream system.

5 Conclusions and future work

Electrical machines with high power densities require intensive cooling methods. Stator windings are often the target for heat extraction, as it is preferential to remove the losses at the source. One method for intensive cooling is to utilise submerged oil jets for targeted fluid impingement on the stator end windings in a fully flooded fluid arrangement. The design and analysis of this cooling method requires high accuracy, due to the strong link between the geometry, fluid flows, heat removal and electrical losses. The CFD methodologies have been described and the fluid and thermal performance of the final experimental testing enclosure for oil cooling of stator components, as well as parametric sensitivity analyses used to develop the design, have been presented.

Some considerations for the design of the oil flow test enclosure are highlighted, such as system pressure distributions. Each component of the system from jets, plenum and delivery pipes have been evaluated for their impact on the pressure. Sensitivity studies for the plenums and oil delivery pipes have been shown. Narrow piping has been shown to provide excessive pressure drops. Piping has been sized to be at a diameter less sensitive for the resulting pressure. The presented system does not require detailed regulation as a pressure vessel, but this must be considered. The CFD provided confidence for the expected pressures and has provided resulting fluid forces to be considered in the structural design. In this case, the CFD was used to ensure a maximum pressure, dictated by the design requirements, was not exceeded. This also ensures the test equipment will operate within the envelope of the laboratory equipment.

Sensitivity studies quantify the behaviour of the jet impact on end-winding surface heat transfer coefficient. This is due to the higher impinging oil velocity. Both heat transfer and peak pressure increase with fluid velocities. As there is a strict pressure limit in this case, the CFD modelling has been used to find the balance between surface heat transfer coefficient and peak pressure, for the most effective cooling.

Future work will reflect commissioning of the experimental test enclosure for the studying of oil cooling of electrical machine components. Sensitivity analyses may be carried out both as independent experimental studies, as well as to validate the CFD work presented here. The CFD modelling presented within this paper will later be fully validated through experimental testing using the test enclosure.

6 References

[1] Wheeler, P.: 'Technology for the more and all electric aircraft of the future'. 2016 IEEE Int. Conf. on Automatica (ICA-ACCA), Curico, 2016, pp. 1–5

[2] Li, Z., Deping, F., Jianhong, G., *et al.*: 'Study on spraying evaporative cooling technology for the large electrical machine'. Electrical Machines and Systems (ICEMS), Tokyo, Japan, 2009

[3] Polikarpova, M.: 'Liquid-Cooling Solutions for Rotating Permanent Magnet Synchronous Machines'. Doctorate thesis, Lappeenranta University of Technology, Finland, 2014

[4] Xu, Z., La Rocca, A., Pickering, S.J., *et al.*: 'Mechanical and thermal design of an aeroengine starter/generator'. IEEE Int. Electric Machines & Drives Conf. (IEMDC), Coeur d'Alene, USA, 2015

[5] Popescu, M., Staton, D., Boglietti, A., *et al.*: 'Modern heat extraction systems for electrical machines – a review'. IEEE Workshop on Electrical Machines Design, Control and Diagnosis (WEMDCD), Torino, 2015, pp. 289–296

[6] Molana, M., Banooni, S.: 'Investigation of heat transfer processes involved liquid impingement jets: a review', *Braz. J. Chem. Eng.*, 2013, **30**, (3), pp. 413–435

[7] Ye, L., Tao, F., Wei, S., *et al.*: 'Experimental research on the oil cooling of the end winding of the motor'. IEEE Energy Conversion Congress and Exposition (ECCE), Milwaukee, WI, 2016, pp. 1–4

[8] John, H., Lienhard, V.: 'Heat transfer by impingement of circular free-surface liquid jets'. 7th ISHMT-ASME Heat and Mass Transfer Conf., India, 2006

[9] Ponomarev, P., Polikarpova, M., Pyrhönen, J.: 'Thermal modeling of directly-oil-cooled permanent magnet synchronous machine'. XXth Int. Conf. on Electrical Machines, Marseille, 2012, pp. 1882–1887

[10] Bersch, K., Connor, P.H., Eastwick, C.N., *et al.*: 'CFD optimisation of the thermal design for a vented electrical machine'. IEEE Workshop on Electrical Machines Design, Control and Diagnosis (WEMDCD), Nottingham, UK, 2017, pp. 39–44

[11] Connor, P.H., Pickering, S.J., Gerada, C., *et al.*: 'Computational fluid dynamics modelling of an entire synchronous generator for improved thermal management', *IET Electr. Power Appl.*, 2013, **7**, (3), pp. 231–236

[12] Tosetti, M., Maggiore, P., Cavagnino, A., *et al.*: 'Conjugate heat transfer analysis of integrated brushless generators for more electric engines', *IEEE Trans. Ind. Appl.*, 2014, **50**, (4), pp. 2467–2475

[13] Jungreuthmayer, C., Winter, O., Bäuml, T., *et al.*: 'A detailed heat and fluid flow analysis of an internal permanent magnet synchronous machine by means of computational fluid dynamics', *IEEE Trans. Ind. Electron.*, 2012, **59**, (12), pp. 4568–4578