

A review on recent studies in the area of conjugate heat transfer modelling in transformers

Rudarsko-geološko-naftni zbornik
(The Mining-Geology-Petroleum Engineering Bulletin)

DOI: 10.17794/rgn.2026.1.14

Review scientific paper



Alen Cukrov^{1*} , Ivanka Boras¹ , Martin Jurina² , Dino Kovačević² , Branimir Čučić² 

¹ University of Zagreb Faculty of Mechanical Engineering and Naval Architecture, Ivana Lučića 5, 10002 Zagreb, Croatia.

² KONČAR - Distribution and Special Transformers, Ul. Josipa Mokrovića 8, 10000 Zagreb, Croatia.

Abstract

The exact position and the value of the hot-spot temperature in a transformer could be estimated by calculation using computational fluid dynamics. However, this approach requires the usage of significant computational resources. In this paper, a review on some recent investigations on this topic is made, focusing thereby primarily on the investigation in which the field distribution of the dependent variables is taken into consideration in a conjugate heat transfer problem. Still, the determination of the hot-spot temperature in the transformer is an open problem. Furthermore, a review on some fundamental aspects related to the heat transfer in the transformer was made, and some new approaches for the calculation of heat transfer in the transformer were proposed: smoothed-particle hydrodynamics, two-fluid model and the application of T-Flows computational software. Thus, the smoothed-particle hydrodynamics is found as a convenient approach, since it is traditionally related to laminar flow, that is present in the transformer; the two-fluid model significantly saves the computational resources, while the application of the computational software T-Flows is free and extremely powerful for computation of conjugate heat transfer and parallel computation. The conducted research defined the guidelines for the future work in numerical modelling of heat transfer in transformers.

Keywords:

transformer; conjugate heat transfer; computational fluid dynamics; hot-spot temperature

1. Introduction

The computational fluid dynamics (CFD) analysis of electric transformers, such as the ones produced in Končar D&ST, a renowned Croatian transformer manufacturer with internationally recognized high-quality of its devices, requires high computational power and significant time consumption. The slow convergence rates of the CFD algorithms and application of multigrid approach for its enhancement is outlined in Volkov (2024) and references therein.

The reason for conducting such a computational demanding analysis is the information about location and value of the hot-spot temperature of the transformer winding, since if this temperature exceeds a certain limit, the aging of the insulation material is much more pronounced and, hence, the safe operation of the device becomes questionable. Other electric equipment is also the subject of active research in the scientific community using CFD techniques, such as the recent investigation of thermal and flow fields in an electric motor (Iacovano

et al., 2024), but would not be further investigated within the scope of the present paper.

In doing so, the practitioners in industry mostly use standard CFD models that are available in the commercial CFD packages and that mostly require many computational cells that, in turn, necessitate the usage of the aforementioned high computational power and consume significant time. The same holds for other areas of industrial importance, such as aerodynamics (Liu et al., 2023; Qin et al. 2024) or naval hydrodynamics (Tezdogan et al., 2015; Kim and Tezdogan, 2022), where detailed computational meshes composed of millions of cells are also used to obtain the field distributions of the dependent variables. Please note that in both cases, aerodynamics and naval hydrodynamics, a single set of governing equations is used.

However, due to the averaging procedure applied in derivation of two-fluid model and corresponding source terms responsible for interphase transport across the interface, the exact information about the interface is lost and the interface could not be tracked explicitly. To mitigate this, several approaches were proposed in the past, all of them being addressed in the introductory part in Mer et al. (2018).

Among the approaches mentioned therein, we shall outline the work of Štrubelj et al. (2009), where the be-

* Corresponding author: Alen Cukrov
e-mail address: alen.cukrov@fsb.unizg.hr

Received: 17 June 2025. Accepted: 8 October 2025.

Available online: 2 January 2026

haviour of the single-fluid model is established by proper modelling of interphase drag term in the momentum equation that leads to equalisation of the velocities at the interface. By having equal velocities at the interface, the two-fluid model behaves as a single-fluid method, preserving the features of the single-fluid methods, while being computationally less expensive. Another approach that belongs to this group of models is the large-bubble model (LBM), presented in **Denèfle et al. (2015)**, where the momentum transport is established within the same phase, but of different size. This is actually an argument for further exploration of the possibility to apply the two-fluid model in practice in the case of oil flow in a transformer.

More recently, in **Cukrov et al. (2023)** the two-fluid VOF method has been used together with an approach termed “frozen turbulence” has been proposed to solve the transient temperature evolution in a solid during the film boiling phase. In this approach the turbulent kinetic energy (TKE) value is prescribed in the domain but differing in the near wall region from the far field. Hence, the wall adjacent cells possess $k = 0 \text{ m}^2/\text{s}^2$, indicating the laminar flow, while outside of this zone the non-zero TKE value is prescribed.

This is in line with **Uruba (2012)**, where it is stated that the dominant turbulence generation is accomplished within the first 5% of the boundary layer. Being around boundary layer theory, in **Dedov et al. (2010)** the jet theory is applied to study liquid-vapor behaviour during a pipe flow. In addition, the two-fluid model has been found as a promising turbulence modelling approach in **Malikov (2020); Malikov (2021); Malikov (2022)**. An example of a two-fluid model algorithm could be found in the recent study by **Li et al. (2024)**.

The study reviews heat transfer modelling options of mostly power transformers and is organised as follows. In Chapter 2, a literature survey has been made, starting from seminal work by **Torriano et al. (2010)** and more focusing on the recent progress in the field. Some other works published in the meantime have been also covered in this chapter. In Chapter 3, a discussion of some aspects presented in the studied literature has been made and important features of the models were addressed. Furthermore, the relationships with other fields were made and some fundamental topics that were not covered in the studied literature were pointed out. Additionally, a software recommendation was made and the steps for future industrial collaborations were stressed. The paper ends with conclusions, Chapter 4, where important items from the work are briefly summarized and the steps for future investigations were proposed.

2. Literature review

A disc-type power transformer winding has been comprehensively studied by usage of CFD technique by **Torriano et al. (2010)**. The generated heat is a conse-

quence of ohmic and eddy losses in the transformer's windings. The authors examined six different cases involving isothermal flow, prescribed heat flux value, assumption of homogeneity of the disc structure (that the disc is completely made of copper instead of its composition, copper and paper insulation), without buoyancy, and buoyancy modelled using Boussinesq assumption.

The validation case in **Torriano et al. (2010)** (full scale, the last case) involved all the necessary features, with the disc being modelled as a continuum with space dependent physical properties (density, specific heat capacity and thermal conductivity), thus alleviating the need for modelling of a small scale interfaces between the paper insulation and copper blocks that would further increase the computational effort. However, the distinction between the fluid and solid domain has been established by the application of generalised grid interface (GGI) procedure, that is common for moving boundary solutions in turbomachinery (**Yang et al., 2024**). The procedure used to solve the CHT problem may be found, for example, in **Cukrov et al. (2025)**.

It was noted by **Torriano et al. (2010)** that a hot spot temperature could be efficiently estimated both by using the full scale simulation and the one with Boussinesq approximation for buoyant flow; the Boussinesq approximation refers to linear dependence of density with respect to the fluid temperature in the buoyancy term of momentum equation, while the density is kept constant in other terms. Due to low velocities of oil flow, the flow is considered as laminar. The heat source is determined from the electromagnetic theory. The influence of mass flow rate of dielectric fluid, that is, oil, has been justified, showing better performance when higher dielectric fluid's mass flow rates were applied.

The correlations for calculation of Nusselt numbers, that are relevant for heat transfer analysis of a power transformer, are given in **Gościński et al. (2016)**. The authors have conducted the CFD simulation of the power transformer using a commercial CFD package ANSYS CFX and have obtained the maximum temperature that is below the limit proposed by the standard in the field.

A thermal-hydraulic code for estimation of hot-spot temperature has been proposed in **Jaramillo (2020)**. The authors verified the proposed approach by comparison of the simulation results with the ones obtained using commercial CFD code ANSYS Fluent. In the numerical simulation conducted by the author, a conformal computational mesh has been chosen at the solid-liquid interface. Thus, the need for GGI is alleviated. The studied cases involve the solution in orthogonal and cylindrical coordinates, that is, the arrangement of power transformer's windings in a plane or in an axisymmetric domain. The volumetric heat source in both cases, planar and axisymmetric, has been set based on the study by **Torriano et al. (2010)**, where it was determined from electromagnetic theory that has not been presented in their work.

The detailed CFD model of the power transformer's radiator has been proposed in the study by **Zhao (2022)**. The author has considered all heat transfer modes, namely the heat conduction in the radiator's wall, the convection of the fluid medium and to the ambient air and the radiation heat transfer from the outer surfaces of the device to the surrounding surfaces that participate in this mode of the heat transfer. The computational model is validated against experiments for seven different cases. The difference is based on different input parameters: top liquid temperature, ambient temperature and liquid flow rate at the top of the radiator. The outputs, bottom liquid temperature and total heat dissipation, were compared to the experimental data.

A comprehensive review on power transformer's conjugate heat transfer processes is given in **Smolyanov et al. (2024)**. As a main source of thermal energy losses the high and low voltage windings were identified, contributing, as per authors, 97% in total thermal energy losses. The authors address relevant methodologies in the investigation of heat transfer in a power transformer, thus noting the electric circuit analogy, thermal network model, and CFD models, the latter being in the focus in the present paper. In solving electromagnetism, the authors emphasise the role of finite element method (FEM) in performing this task. This may be explained with the fact that structural modelling, in which the FEM is usually used, is similar to electromagnetism in a sense that there is no viscosity, and hence no advection that is usually solved using finite volume method (FVM).

The possibility of solving the heat transfer within power transformer winding using correlations for heat transfer coefficient has been also outlined in the work by **Smolyanov et al. (2024)**. Thus, one alleviates the need to solve the liquid part of the domain, which may lead to discrepancies in the simulation result. In addition, the role of the Richardson number in determination of hot-spot temperature has been also outlined. Furthermore, the different correlations for the estimation of the Nusselt number were shown, being divided in one suitable for a large-scale domain, i.e. the large transformer, that are derived from boundary layer theory for flat plates, and those appropriate for small-scale domains. Some of these correlations slightly differ from those mentioned in the work by **Gościński et al. (2016)**.

The influence of uneven distribution of electromagnetic losses on the temperature field in a dry-type iron-core reactor has been studied in **Liao et al. (2024)**. A multiphysics simulation has been carried out for a 10 kV reactor in a coupled electromagnetic, thermal and flow field simulation. Firstly, an electromagnetic simulation has been carried out, and the orders of eddy and ohmic losses were determined. This result is further transferred for the purposes of the thermal simulation in which the temperature field has been obtained. Then, a fluid flow field has been solved and the vice versa approach has been utilized. The natural convection cooling is estab-

lished within an enclosed space, thus simulating, at least to a certain extent, the real situation in which this type of reactor is usually placed.

The reduced order modelling (ROM) influenced by a full order model (FOM) in estimation of 3D temperature field within transformer winding is presented in **Liu et al. (2024)**. The validation of the approach has been conducted using CFD and measurement data on a 35 kV power transformer winding immersed in oil, and the significant time savings were reported in obtaining the temperature field.

A detailed analysis of a power transformer using wedge-type geometry is conducted in **Wang et al. (2024)**. In the study, a hex-dominant computational mesh composed of 24 million cells is used and all the heat transfer modes were taken into consideration. The tank with corrugated wall surface, that is difficult to handle with thermal-hydraulic models (THM), has been studied using CFD technique.

The CFD analysis aimed at determining a hot-spot temperature in an oil cooled power transformer has been carried out using computational software ANSYS Fluent in **Taghikhani (2024)**. In addition, the spatial distribution of the magnetic field has been obtained. Furthermore, within the CFD study, the author has conducted a mesh sensitivity check, and the comprehensive CFD simulation using the computational mesh composed of several million cells. The results were compared with the grey wolf optimisation (GWO) algorithm proposed by the author, and a close correspondence with the CFD result has been achieved. In the conducted CFD simulation the heat transfer coefficient has been prescribed at the outer walls of the tank.

The dynamic heat transfer coefficient approach has been used in study by **Xiao et al. (2024)**. The heat transfer coefficient was determined using a classical approach. The infrared thermography has been applied to determine the dynamic heat transfer coefficients that were used to obtain the exact heat transfer coefficient. A 3D numerical simulation has also been carried out with detailed insight in the flow and thermal fields.

This review is completed with the recent work by **Lin et al. (2025)**, who have reviewed the techniques for determination of the hot spot temperature in an oil-immersed power transformer. In their study, firstly three different losses that lead to heat generation (iron-core, winding and stray losses) were identified, and methods for their calculation were discussed. Then, the problem of hot-spot temperature estimation has been tackled from the viewpoint of two distinct approaches: measurement and calculation. In the latter approach, four different calculation methods were identified: empirical calculation, thermal circuit modelling, numerical simulations and artificial intelligence (AI).

In their work, **Lin et al. (2025)**, the presentation of hot-spot temperature dependency on the transformer height is conducted by a linear function and is identified

as transformer type dependent. Among the calculation methods for estimation of hot-spot temperature, numerical simulations and AI were identified as the ones that can accurately estimate the location of the hot-spot temperature.

However, as per **Lin et al. (2025)**, the numerical simulations are considered very slow and computational demanding, while AI techniques depend on training of the model. Furthermore, the features of oil flow were related to the hot-spot temperature. For example, it was found that the increased oil flow rate at the inlet influences the decrease in the hot-spot temperature. The number of baffles, furthermore, has to be specified with care since its increase results in a decrease of the hot-spot temperature to a certain level. After this level is reached, further addition of baffles would yield an increase in the hot-spot temperature.

3. Discussion

In this section, the studied literature is further discussed and the relations with other fields were made. Furthermore, some aspects that were not found in the studied literature were identified and discussed. Possibilities for industrial cooperation based on findings is given at the end of this chapter.

3.1. Discussion on the studied literature with the relations to other fields

Since the seminal work of **Torriano et al. (2010)** the classical computational strategies were, to the authors' knowledge, found as the ones that are used mostly to determine the location of the hot-spot temperature in the windings of the oil-immersed power transformer in application of CFD techniques. This refers to mesh based CFD methods using standard continuum mechanics features. Furthermore, the work by **Torriano et al. (2010)**, in which the winding temperature distribution has been successfully estimated using the Boussinesq approach, may be, at least to a certain extent, related to the work by **Dović (2005)**, who has investigated the flow behaviour in a solar collector. In his study, the author has applied the Boussinesq model to solve the buoyancy driven flow in the device.

The idea of prescription of the heat transfer coefficient at the domain's boundaries in order to avoid modelling of the fluid medium is shown in **Smolyanov et al. (2024)**, referring to the work by **Doolgindachbaporn et al. (2022)**. This approach is, however, used in numerical modelling of heat treatment, e.g. quenching, where the CFD-determined heat transfer coefficient is used for further analyses within material in the recent article by **Gamidi and Passam (2024)**.

The natural convection cooling of a dry-type iron-core transformer reported in **Liao et al. (2024)** follows similar trends as in the case of natural convection around

a square shown in the work by **Savio et al. (2022)**. In addition, the natural convection around a differentially oriented square has been studied in **De and Dalal (2006)**. In both cases one may note the increased velocity at the upper parts of the object in a buoyancy driven flow. In the area of determination of eddy current losses, that were determined together with ohmic losses to determine the temperature field, we may also outline the recent work by **Frlić (2023)**. The author has conducted the estimation of the eddy current losses using finite element method.

It is interesting to note that in the studied literature it was found that the problem of natural convection in enclosures with differentially heated walls has not been tackled in sufficient detail. The conceptual sketch of this problem is shown in the left part of **Figure 1**. It is about the buoyancy driven flow induced by the different temperatures of side walls as shown, for example in **Wu (2015)**.

A comprehensive review on natural convection in enclosures is given in **Miroshnichenko and Sheremet (2018)**, while a more recent review by **Rashid et al. (2022)** also includes the presence of nanofluids in the flow. A theoretical treatise on the subject, together with correlations for the calculation of the Nusselt number is given in **Abramov et al. (2020)**, while the further enhancement of the topic by consideration of conjugate heat transfer could be found, for example, in **Bilgen (2009)** and is schematically shown in the right part of **Figure 1**.

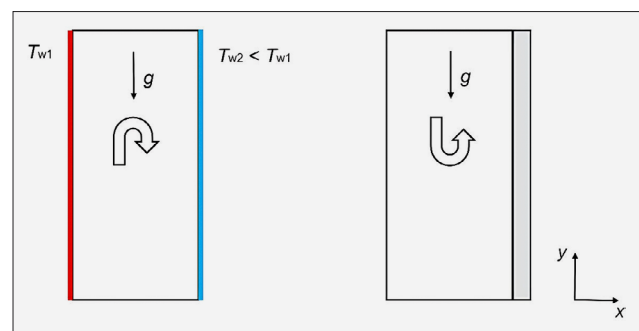


Figure 1. A conceptual sketch of natural convection in cavity: (a) without conjugate heat transfer; (b) with conjugate heat transfer. Image enhanced using PhotoGrid software (<https://www.photogrid.app/en/>)

The radial basis function (RBF) technique, that is used in conjunction with neural network (NN) in an RB-FNN approach, mentioned in **Liu et al. (2024)**, is also an acknowledged approach in handling complex mesh motion in CFD modelling. See **Niu et al. (2017)** for more details on the approach. Other examples of usage of this technique in mesh deformation handling are available in **de Boer et al. (2007)**. The RBF, furthermore, is mentioned as one of the relevant metamodelling in the recent study on aerodynamics shape optimization in **Zhang et al. (2024)**.

The application of optimisation algorithms to reduce the consumption of computational resources, such as in the case of GWO algorithm usage in **Taghikhani (2024)**, could be related to weather forecast, where AI could be used to estimate the weather conditions, shown in the lecture by **Güttler (2024)**. However, although faster, it is significantly less accurate since the accuracy, as noted by the author, lies in the physics.

In the conducted literature survey, one may note that following topics were not addressed by the authors so far and that would be discussed in more detail in the following text.

- The application of particle based smoothed-particle hydrodynamics (SPH) approach that, at present, is applied in the automotive industry for solving complex tasks.
- The application of the two-fluid model has not been found.
- The application of T-Flows, an open-source CFD software capable for flow modelling including heat transfer, has not been used by any author in the studied literature.

The engineering application of the discussion outcomes is discussed later, after the above three items are presented in more detail.

3.2. The application of SPH

In SPH, the continuum is discretized using a finite number of particles (Lagrangian approach) instead of computational cells (Eulerian approach) used in mesh based CFD. The brief exposition of the method fundamentals could be found, for example, in recent works (**Pozorski and Olejnik, 2024; Sigalotti and Vargas, 2025; Le Touzé and Colagrossi, 2025**). The method has difficulties with boundary handling, turbulent flow modelling and requires huge computational resources. An approach for handling boundaries is recently proposed in **Kim and Torrens (2024)**.

The other issue is tensile instability that could be remedied using the energy stable schemes, as presented in **Zhu (2024)**. However, it has shown great potential for handling complex geometries, such as those in the automotive industry where the lubrication flow together with conjugate heat transfer (CHT) is solved in **Klos et al. (2024)**. In their presentation, the fluid flow is modelled using SPH, while the coupling with commercial CFD software is made for purposes of solving the heat conduction equation in the solid part of the domain.

Other applications in the automotive sector include the sensor pollution simulation, presented in **Kleineberg and Singh (2024)** and car motion through liquid-water (motion through deep water and fast motion through water), shown in **Posch (2024)**. From the scientific point of view, we may outline the complex fluid-structure interaction (FSI) problem in a shock-wave induced load on a structure, presented in **Chen et al. (2024)**. The authors

model the fluid part using the finite-difference method (FDM), since it is more suitable for modelling the compressible flow features studied in the paper, while the motion of the solid part is modelled using the SPH.

The coupling between FDM and SPH, shown in **Chen et al. (2024)** is established using the immersed boundary method (IBM), which is also an approach in conjugate heat transfer analysis of metal quenching in **Kosseifi (2012)**. Since the flow in the transformer is considered as laminar (**Zhao, 2022**), the difficulty in handling turbulent motion associated with the method, may not be regarded as a problem. The application of SPH in solving natural convection in a differentially heated square cavity is presented in **Yang et al. (2021)**.

3.3. The application of two-fluid model

In the application of the large bubble model (LBM), described in **Denèfle et al. (2015)**, the interfacial momentum transfer is accomplished within the same phase, but with a different size of the interface. Thus, one may model various scales in a two-fluid four field approach, which leads to reasonable consumption of computational time (**Gauss et al., 2016**) and explicitly stated in **Pointer and Liu (2017)**.

Using the two-fluid VOF model in conjunction with frozen turbulence model is applied to solve the conjugate heat transfer during film boiling around a silver cylinder in **Cukrov et al. (2023)** and the method for estimation of the turbulent kinetic energy estimation based on Kelvin-Helmholtz instability (KHI) theory model proposed in **Hillier et al. (2020)** has been proposed.

The foundations of the approach proposed in **Cukrov et al. (2023)** is depicted in **Figure 2** and the underlying modelling approach is described in **Cukrov et al. (2021)**, while its application in temperature and heat flux estimation during the quenching process is demonstrated in **Cukrov et al. (2023a)** and **Cukrov et al. (2024)**, respectively. The complete development of the method is presented in **Cukrov et al. (2025a)**.

Regarding the KHI, it is a flow mode that appears in a single-phase flow, such as the jet flow where the shear is induced by different velocities of the jet issuing from the nozzle and the quiescent surrounding medium. In addition, the phenomenon may also be induced in a two-phase flow, where the phases flow with different velocities and thus induce the shear if the relative velocity exceeds a certain velocity limit described in a recent study by **Krpan et al. (2023)**.

By involving turbulent kinetic energy value using KHI, we follow the so-called “physics informed” approach, but with a difference from the standard one in which the information is gathered from the comprehensive set of data, we close the modelling approach using the data obtained from theory. This was recently applied in a study by **Jang et al. (2024)** where a Timoshenko beam theory has been involved in structural analysis using discrete element method (DEM).

The involvement of the dispersed formulation of a realizable k - ε turbulence model within a two-fluid VOF model available in ANSYS Fluent allows for definition of laminar zone in the vicinity of solid surface (laminar zone in the left part of **Figure 2**) with $k \cong 0 \text{ m}^2/\text{s}^2$, while the remainder of the domain is filled with a finite value of turbulent kinetic energy, i.e. $k > 0 \text{ m}^2/\text{s}^2$.

As the flow evolves, as shown in the right part of **Figure 2**, the laminar zone is preserved while the turbulent zone is present only outside the jet structure since the turbulent viscosity is a function of strain rate tensor and, hence, the turbulence occurs on the interface between the zones and in the complete liquid phase. This could be related to the assumption in the work of **Dedov et al. (2010)**, where jet theory is used to this end, but with the difference that in their work, a pipe flow is considered.

An analogy with the jet flow could be made because in this type of flow, the maximum turbulence intensities are on the outer surfaces of the jet, as shown in the experimental work by **Philip (1972)**. Hence, even in the single-phase flow, we may apply this approach by assigning the zero turbulent kinetic energy value in the vicinity of the stationary walls, while keeping it non-zero out of this area.

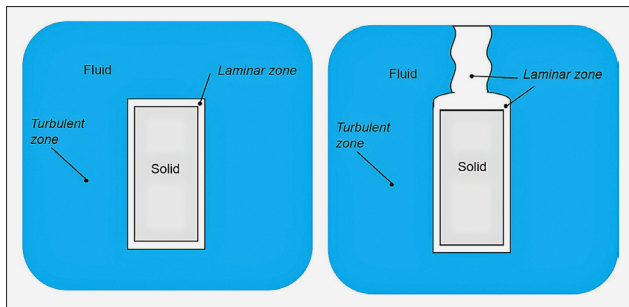


Figure 2. The proposed two-fluid VOF modelling concept: (a) initial state; (b) evolved flow. Image enhanced using PhotoGrid software (<https://www.photogrid.app/en/>).

3.4. The application of T-Flows open-source CFD code

T-Flows is an open-source computational software capable for parallel computation of heat transfer and fluid flow, and is proposed in **Ničeno (2001)**. Furthermore, the software also covers advanced flow modelling; the software tutorial is available at **Sayed (2024)**. Its convenience in solving environmental aerodynamics problems has been shown in **Hadžiabdić et al. (2022)**. In doing so, an automatic mesh generation procedure using GMSH (**Geuzaine and Remacle, 2009**), an open-source mesh generator, has been proposed.

Furthermore, it was also successfully applied in solving the Rayleigh-Bernard convection in an enclosure (**Hadžiabdić et al., 2021**). The advanced turbulence modelling options were demonstrated in heat transfer and fluid flow modelling in **Palkin et al. (2021)**; **Palkin**

et al. (2016); **Palkin et al. (2018)**; **Palkin and Mullyadzhanov (2019)**. The application of this software in the transformer modelling has a great potential since the software is freely available and has parallel computation capabilities between any header of different ranks.

3.5. The possibilities for application of two-fluid model in product development sector of Končar D&ST

The high visibility of Končar D&ST products worldwide stems from the high-quality engineering that is incorporated in the R&D sector of the company. The advanced approaches applied to this end may be tracked from works (**Marković and Čučić, 2018**; **Meško et al., 2020**; **Čučić et al., 2014**). Within the modelling and simulation group of the company, highly skilled professionals (**Kovačević et al., 2024**) with a strong background in thermodynamics, fluid mechanics and heat transfer (the thermofluids) conduct the temperature and flow field analyses of a transformer as well as the electromagnetic analysis.

This work serves as a basis for future work on application of the two-fluid model in transformer development, established between Končar D&ST and University of Zagreb's Faculty of Mechanical Engineering and Naval Architecture. The reason for this is the great potential of the method that has been recognized in the research and development team in Končar D&ST.

4. Conclusions

The hot-spot temperature in a transformer is to appear at the top part of the device and was successfully determined by using computational techniques by the scholars whose publications were studied within this research. Furthermore, the relations with other areas are made and some aspects that were not found in the studied literature were revealed, and the following conclusions may be drawn.

- The full order models of the transformers yield accurate results, but at a high computational cost and significant time consumption.
- The usage of optimisation techniques may lead to the achievement of accurate results without significant time consumption; however, the inexistence of underlying continuum mechanics fundamentals in such approaches may be found as a loss of generality of such approaches.
- AI algorithms produce faster solutions, but since the true accuracy stems from the appropriate modelling of underlying physics, should be used with care.
- Although very popular in the automotive industry, the application of SPH may yield some other problems, such as boundary handling and high computational cost.

- The application of novel approaches, in particular the two-fluid VOF method, may lead to a significant decrease in the consumption of computational resources and time.
- The strong parallel computation capabilities of the open-source CFD software T-Flows may be applied for future calculations of the hot-spot temperatures in the transformers.
- The engineering practitioners in industry have recognized the potential of application of the two-fluid modelling in the case of single-phase flow using theory based turbulence assumptions.

Finally, we can conclude that still more work has to be carried out in order to minimise the computational effort required for determination of the hot-spot temperature. The future steps include fundamental research on the two-fluid modelling of single-phase flow using frozen turbulence approach. Hence, the turbulence (or the laminar flow) quantities should stem from careful theoretical reasoning rather than statistics.

5. References

- Abramov, A.G., Ivanov, N.G., Ris, V.V. (2020). Fluid flows in the fields of volume forces. Laminar regimes and stability of free-convective flows. Peter the Great St. Petersburg Polytechnic University (in Russian). https://aero.spbstu.ru/attachments/article/81/abramov-ivanov-ris-free_convection.pdf
- Bilgen, E. (2009). Conjugate heat transfer by conduction and natural convection on a heated vertical wall, *Applied Thermal Engineering*, 29, 2–3, 334–339, <https://doi.org/10.1016/j.applthermaleng.2008.02.038>
- Chen, J.-Y., Feng, D.-L., Peng, C., Ni, R.-C., Wu, Y.-X., Li, T., Song, X.-Z. (2024). A coupled FD-SPH method for shock-structure interaction and dynamic fracture propagation modelling, *Applied Mathematical Modelling*, 134, 288–306. <https://doi.org/10.1016/j.apm.2024.06.010>
- Cukrov, A., Sato, Y., Boras, I., Ničeno, B. (2023). Film Boiling around a Finite Size Cylindrical Specimen – A Transient Conjugate Heat Transfer Approach. *Applied Sciences* 13. <https://doi.org/10.3390/app13169144>
- Cukrov, A., Sato, Y., Boras, I., Ničeno, B. (2021). A solution to Stefan problem using Eulerian two-fluid VOF model. *Brodogradnja : Teorija i praksa brodogradnje i pomorske tehnike* 72, 141–164. <https://doi.org/10.21278/brod72408>
- Cukrov, A., Landek, D., Sato, Y., Boras, I., Ničeno, B. (2023a). Water Entry of a Heated Axisymmetric Vertical Cylinder. *Energies* 16. <https://doi.org/10.3390/en16247926>
- Cukrov A., Landek D., Sato Y., Boras I., Ničeno B. (2024). Numerical simulation of film boiling heat transfer in immersion quenching process using Eulerian two-fluid approach. *Case Studies in Thermal Engineering*. 64:105497. doi: 10.1016/j.csite.2024.105497
- Cukrov A., Sato Y., Landek D., Hannoschöck N., Boras I., Ničeno B. (2025). Determination of Heat Transfer Coefficient in a Film Boiling Phase of an Immersion Quenching Process. *Applied Sciences*. 15(3):1021. <https://doi.org/10.3390/app15031021>
- Cukrov A., Landek D., Sato Y., Boras I., Ničeno B. (2025a). The development of the numerical method for simulation of metal material quenching. *Rudarsko-geološko-naftni zbornik* 40(1):29-50. <https://doi.org/10.17794/rgn.2025.1.3>
- Ćučić, B., Meško, N., Mikulić, M., Trstoglavac, D. (2014). Design improvements in modern distribution transformers, *Journal of Energy*, vol. 63 Number 1–4, Special Issue, p. 74–81 <https://doi.org/10.37798/2014631-4165>
- De, A.K., Dalal, A. (2006). A numerical study of natural convection around a square, horizontal, heated cylinder placed in an enclosure, *International Journal of Heat and Mass Transfer*, 49, 23–24, 2006, 4608–4623, <https://doi.org/10.1016/j.ijheatmasstransfer.2006.04.020>
- de Boer, A., van der Schoot, M.S., Bijl, H. (2007) Mesh deformation based on radial basis function interpolation, *Computers & Structures*, 85, 11–14, 2007, 784–795, <https://doi.org/10.1016/j.compstruc.2007.01.013>
- Dedov, A.V., Komov, A.T., Varava, A.N., Yagov, V.V. (2010). Hydrodynamics and heat transfer in swirl flow under conditions of one-side heating. Part 2: Boiling heat transfer. Critical heat fluxes, *International Journal of Heat and Mass Transfer*, 53, 21–22, 4966–4975, <https://doi.org/10.1016/j.ijheatmasstransfer.2010.05.035>
- Deněfle, R., Mimouni, S., Caltagirone, J.-P., Vincent, S. (2015). Multifield hybrid approach for two-phase flow modelling – Part I: Adiabatic flows, *Computers & Fluids*, 113, 106–111. <https://doi.org/10.1016/j.compfluid.2014.07.018>
- Doolgindachbaporn, A., Callender, G., Lewin, P.L., Simonson E. and Wilson, G. (2022). A Top-Oil Thermal Model for Power Transformers That Considers Weather Factors, in *IEEE Transactions on Power Delivery*, vol. 37, no. 3, pp. 2163–2171, June 2022, doi: 10.1109/TPWRD.2021.3105459
- Dović, D. (2005). Numerička i eksperimentalna studija sunčevog kolektora s valovitim apsorberom, Doctoral thesis, University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, Zagreb, 2005. Available at: <https://urn.nsk.hr/urn:nbn:hr:235:316672>
- Frljić, S. (2023). Izračun gubitaka zbog vrtložnih struja u lameliranoj transformatorskoj jezgri otvorenoga tipa primjenom metode konačnih elemenata, Doctoral thesis, University of Zagreb, Faculty of Electrical Engineering and Computing, Zagreb. Available at: <https://urn.nsk.hr/urn:nbn:hr:168:593565>
- Gamidi, K., Pasam, V.K. (2024) Analysis of Residual Stresses in Spot Cooled Vibration Assisted Turning of Ti6Al4V Alloy Using Computational Fluid Dynamics-Aided Finite Element Method. *Journal of Materials Engineering and Performance*. 33, 3731–3745. <https://doi.org/10.1007/s11665-023-08702-x>
- Gauss, F., Lucas, D., Krepper, E. (2016). Grid Studies for the Simulation of Resolved Structures in an Eulerian Two-Fluid Framework. *Nuclear Engineering and Design*. 305, 371–377. <https://doi.org/10.1016/j.nucengdes.2016.06.009>
- Geuzaine, C., Remacle, J.-F. (2009). Gmsh: A 3-D Finite Element Mesh Generator with Built-in Pre- and Post-Processing Facilities. *International Journal for Numerical Meth-*

- ods in Engineering. 79, 1309–1331. <https://doi.org/10.1002/nme.2579>.
- Gościński, P., Dombek, G., Nadolny, Z. and Bródka, B. (2016) Matematyczne modelowanie pola temperaturowego transformatora energetycznego. *Poznan University of Technology Academic Journals. Electrical Engineering*. <https://bibliotekanauki.pl/articles/376090.pdf>
- Güttler, I. (2024). Hidrometeorološke usluge i servisi nove generacije. Od ranog upozorenja do klimatske neutralnosti. Srce DEI 2024. https://www.srce.unizg.hr/arhiva_weba/dei/04112024/dei.srce.hr/2024/prezentacije.html
- Hadžiabdić, M., Hafizović, M., Ničeno, B., & Hanjalić, K. (2022). A rational hybrid RANS-LES model for CFD predictions of microclimate and environmental quality in real urban structures. *Building and Environment*, 217, Article 109042. <https://doi.org/10.1016/j.buildenv.2022.109042>
- Hadžiabdić, M., Borrelli, M. and Ničeno, B. (2021). Simulation of Rayleigh-Benard convection at up to $Ra=1016$ by generalized elliptic-relaxation hybrid RANS-LES model, *International Journal of Heat and Fluid Flow*, 90, 108819, <https://doi.org/10.1016/j.ijheatfluidflow.2021.108819>
- Hillier, A., Doorselaere, T.V., Karampelas, K. (2020) Estimating the Energy Dissipation from Kelvin–Helmholtz Instability Induced Turbulence in Oscillating Coronal Loops. *ApJL*, 897, L13. 10.3847/2041-8213/ab9ca3
- Iacovano, C., Berni, F., Cicalese, G., Nuzzo, S., Fontanesi, S. (2024). An integrated 2D/3D numerical methodology to predict the thermal field of electric motors, *Case Studies in Thermal Engineering*, 56, 104233, <https://doi.org/10.1016/j.csite.2024.104233>
- Jang, Y., Kim, E., Yang, J., Rho, J. (2024). Physics-informed discrete element modelling for the bandgap engineering of cylinder chains, *Applied Mathematical Modelling*, 125, Part A, 571-590, <https://doi.org/10.1016/j.apm.2023.09.011>
- Jaramillo, G. A. P. (2020). Modelización termo-hidráulica del proceso de refrigeración de transformadores de potencia bañados en aceite, Doctoral dissertation, Universidad Carlos III de Madrid. <https://portalinvestigacion.upct.es/documentos/6381691618a84b178fea9e5e>
- Kim, D. and Tezdogan, T. (2022). CFD-based hydrodynamic analyses of ship course keeping control and turning performance in irregular waves, *Ocean Engineering*, 248, 110808, <https://doi.org/10.1016/j.oceaneng.2022.110808>.
- Kim, R., & Torrens, P. M. (2024). Boundary SPH for Robust Particle–Mesh Interaction in Three Dimensions. *Algorithms*, 17(5), 218. <https://doi.org/10.3390/a17050218>
- Kleineberg, S. and Singh, Y. (2024). Determination of the real sensor pollution for the advanced sensor cleaning system of the vehicle. 2024, AVL Simulation Conference Germany 2024. 24.04.2024. Regensburg, Germany. <https://www.avl.com/de-de/downloads-avl-simulation-conference-germany-2024>
- Klos, F., Hoffmann, M. and Rothe, S. (2024). Conjugate Heat Transfer Simulation mittels PreonLab. AVL Simulation Conference Germany 2024. 24.04.2024. Regensburg, Germany. <https://www.avl.com/de-de/downloads-avl-simulation-conference-germany-2024>
- El Kousseifi, N. (2012) Numerical Simulation of Boiling for Industrial Quenching Processes, Doctoral dissertation.
- Ecole Nationale Supérieure des Mines de Paris: Paris, France. <https://theses.hal.science/pastel-00734601/>
- Kovačević, D., Marković, M. & Blažina, D. (2024). Numerical Calculation and Direct Measurement of Local Hot Spot Temperatures in Transformer Clamping Plates. *Journal of Energy*, 73.(2.), 8-12. <https://doi.org/10.37798/2024732513>
- Krpan, R., Tiselj, I. and Kljenak, I. (2023). Atmosphere homogenization induced by vertical jets in large enclosures, *Annals of Nuclear Energy*, 180, 109476, <https://doi.org/10.1016/j.anucene.2022.109476>.
- Le Touzé, D. and Colagrossi, A. (2025) Smoothed particle hydrodynamics for free-surface and multiphase flows: a review. *Reports on Progress in Physics*, 88 (3), 037001, <https://dx.doi.org/10.1088/1361-6633/ada80f>
- Li, X., Tian, R., He, L., Lv, Y., Zhou, S. and Li, Y. (2024). A novel numerical approach for assessing the gas-liquid flow characteristics in pipelines utilizing a two-fluid model, *Applied Mathematical Modelling*, 131, 233-252, <https://doi.org/10.1016/j.apm.2024.04.006>.
- Liao, C., Liu, B., Li, K. and Wang, W. (2024). Accuracy improvement of thermal distribution analysis of 10 kV dry-type iron-core reactor, *Case Studies in Thermal Engineering*, 64, 2024, 105554, <https://doi.org/10.1016/j.csite.2024.105554>
- Lin, L., Qiang, C., Zhang, H., Chen, Q., An, Z., & Xu, W. (2025). Review of Studies on the Hot Spot Temperature of Oil-Immersed Transformers. *Energies*, 18(1), 74. <https://doi.org/10.3390/en18010074>
- Liu, Z., Soper, D., Hemida, H. and Chen, B. (2023). A study of the influence of separation bubbles around a generic freight train on pressure waves inside tunnels using 1D and 3D numerical methods, *Journal of Wind Engineering and Industrial Aerodynamics*, 240, 105461, <https://doi.org/10.1016/j.jweia.2023.105461>.
- Liu, Y., Zhao, Q., Liu, G., Zou, Y., Zhang, S., Wang, K. and Zhao, X. (2024) A real-time solution method for three-dimensional steady temperature field of transformer windings based on mechanism-embedded cascade network, *Case Studies in Thermal Engineering*, 58, 104444, <https://doi.org/10.1016/j.csite.2024.104444>
- Malikov, Z. (2020). Mathematical model of turbulence based on the dynamics of two fluids, *Applied Mathematical Modelling*, 82, 409-436, <https://doi.org/10.1016/j.apm.2020.01.047>.
- Malikov, Z.M. (2021). Mathematical model of turbulent heat transfer based on the dynamics of two fluids, *Applied Mathematical Modelling*, 91, 186-213, <https://doi.org/10.1016/j.apm.2020.09.029>.
- Malikov, Z.M. (2022). Modelling a turbulent multicomponent fluid with variable density using a two-fluid approach, *Applied Mathematical Modelling*, 104, 2022, 34-49, <https://doi.org/10.1016/j.apm.2021.10.045>
- Marković, M. & Čučić, B. (2018). Transformer Lifetime Management through Solid and Liquid Insulation Assessment. https://www.researchgate.net/publication/330925194_Transformer_Lifetime_Management_through_Solid_and_Liquid_Insulation_Assessment
- Mer, S., Praud, O., Neau, H., Merigoux, N., Magnaudet, J. and Roig, V. (2018). The Emptying of a Bottle as a Test Case

- for Assessing Interfacial Momentum Exchange Models for Euler–Euler Simulations of Multi-Scale Gas-Liquid Flows. *International Journal of Multiphase Flow*, 106, 109–124. <https://doi.org/10.1016/j.ijmultiphaseflow.2018.05.002>
- Meško, N., Čučić, B., & Žarko, D. (2020). Critical stress of oval foil winding with epoxy coated insulation determined using measured equivalent modulus of elasticity. *Automatika*, 61(3), 388–395. <https://doi.org/10.1080/00051144.2020.1761100>
- Miroshnichenko, I.V. and Sheremet, M.A. (2018). Turbulent natural convection heat transfer in rectangular enclosures using experimental and numerical approaches: A review, *Renewable and Sustainable Energy Reviews*, 82, Part 1, 40–59, <https://doi.org/10.1016/j.rser.2017.09.005>.
- Ničeno, B. (2001). An Unstructured Parallel Algorithm for Large Eddy and Conjugate Heat Transfer Simulations, PhD Thesis, TU Delft. <https://resolver.tudelft.nl/ft1280820-20ce-4c5f-bd0b-8f70e2c2e965> Software available at: <https://github.com/DelNov/T-Flows>
- Niu, J., Lei, J., He, J. (2017). Radial basis function mesh deformation based on dynamic control points, *Aerospace Science and Technology*, 64, 122–132, <https://doi.org/10.1016/j.ast.2017.01.022>
- Palkin, E., Mullyadzhannov, R., Hadžiabdić, M. et al. (2016). Scrutinizing URANS in Shedding Flows: The Case of Cylinder in Cross-Flow in the Subcritical Regime. *Flow Turbulence Combust* 97, 1017–1046. <https://doi.org/10.1007/s10494-016-9772-z>
- Palkin, E. and Mullyadzhannov, R. (2019). Influence of stream-wise vortical structures on heat transfer in the far cylinder wake in a slot channel flow, *Journal of Physics: Conference Series*. 1359 012011. [10.1088/1742-6596/1359/1/012011](https://doi.org/10.1088/1742-6596/1359/1/012011)
- Palkin, E., Mullyadzhannov, R., Hadziabdic, M. and Hanjalic, K. (2021). Active heat transfer and flow control over a cylinder by rotary oscillations. *AIP Conf. Proc.*; 2351 (1): 040044. <https://doi.org/10.1063/5.0053130>
- Palkin E, Hadžiabdić M, Mullyadzhannov R, Hanjalić K. (2018). Control of flow around a cylinder by rotary oscillations at a high subcritical Reynolds number. *Journal of Fluid Mechanics*. 855:236-266. [doi:10.1017/jfm.2018.639](https://doi.org/10.1017/jfm.2018.639)
- Philip, G. (1972). Measurement of turbulence in an annular jet. University of Missouri--Rolla. Masters Theses. 5061. https://scholarsmine.mst.edu/masters_theses/5061
- Pointer, W.D. and Liu, Y. (2017). Eulerian two-fluid RANS-based CFD simulation of a helical coil steam generator boiling tube, in: *Proc. 17th International Topical Meeting on Nuclear Reactor Thermal Hydraulics*. Xi'an, China, p. 19. https://www.researchgate.net/publication/342534046_EULERIAN_TWO-FLUID_RANS-BASED_CFD_SIMULATIONS_OF_A_HELICAL_COIL_STEAM_GENERATOR_BOILING_TUBE
- Posch, K. (2024). Validierung von Wasserdurchfahrtsimulationen. *AVL Simulation Conference Germany 2024*, Regensburg, Germany. <https://www.avl.com/de-de/downloads-avl-simulation-conference-germany-2024>
- Pozorski, J. and Olejnik, M. (2024) Smoothed particle hydrodynamics modelling of multiphase flows: an overview. *Acta Mech* 235, 1685–1714. <https://doi.org/10.1007/s00707-023-03763-4>
- Qin, P., Ricci, A. and Blocken, B. (2024). CFD simulation of aerodynamic forces on the DrivAer car model: Impact of computational parameters, *Journal of Wind Engineering and Industrial Aerodynamics*, 248, 105711, <https://doi.org/10.1016/j.jweia.2024.105711>.
- Rashid, F. L., Hussein, A. K., Malekshah, E. H., Abderrahmane, A., Guedri, K., & Younis, O. (2022). Review of Heat Transfer Analysis in Different Cavity Geometries with and without Nanofluids. *Nanomaterials*, 12(14), 2481. <https://doi.org/10.3390/nano12142481>
- Savio, R.R., Shaik, S. & Kumar, R.S. (2022). Numerical study of natural convection around a square cylinder within a square enclosure for different orientations. *J Therm Anal Calorim* 147, 1711–1725, <https://doi.org/10.1007/s10973-020-10499-z>
- Sigalotti L.D.G. and Vargas C.A. (2025). Smoothed particle hydrodynamics (SPH) simulations of drop evaporation: A comprehensive overview of methods and applications. *Comput Model Eng Sci*.142(3):2281–2337. <https://doi.org/10.32604/cmesci.2025.060497>
- Smolyanov I, Shmakov E, Butusov D, Khalyasmaa AI. (2024). Review of Modelling Approaches for Conjugate Heat Transfer Processes in Oil-Immersed Transformers. *Computation*. 12(5):97. <https://doi.org/10.3390/computation12050097>
- Štrubelj, L., Tišelj, I. and Mavko, B. (2009). Simulations of free surface flows with implementation of surface tension and interface sharpening in the two-fluid model, *International Journal of Heat and Fluid Flow*, 30 (4) 741-750. <https://doi.org/10.1016/j.ijheatfluidflow.2009.02.009>
- Taghikhani, M.A. (2024). Heat transfer modelling in a transformer using magneto-CFD analysis and GWO algorithm considering dynamic thermal model, *Results in Engineering*, 23, 102856, <https://doi.org/10.1016/j.rineng.2024.102856>
- Tezdogan, T., Demirel, Y.K., Kellett, P., Khorasanchi, M., Incecik, A. and Turan, O. (2015). Full-scale unsteady RANS CFD simulations of ship behaviour and performance in head seas due to slow steaming, *Ocean Engineering*, 97, 186–206, <https://doi.org/10.1016/j.oceaneng.2015.01.011>.
- Torriano, F., Chaaban, M. and Picher, P. (2010). Numerical study of parameters affecting the temperature distribution in a disc-type transformer winding, *Applied Thermal Engineering*, 30, 14–15, 2034–2044. <https://doi.org/10.1016/j.applthermaleng.2010.05.004>
- Uruba, V. (2012). *Turbulence Handbook for Experimental Fluid Mechanics Professionals*. Skovlunde: Dantec Dynamic. https://home.zcu.cz/~uruba/vyuka/PIV_HW/mat/Turbulence%20Uruba%20EN_44_aff.pdf
- Volkov, K. (2024). Vectorized Numerical Algorithms to Solve Internal Problems of Computational Fluid Dynamics. *Algorithms*, 17(2), 50. <https://doi.org/10.3390/a17020050>
- Wang, C., Sun, Q., Al-Abadi, A., & Wu, W. (2024). A Combined Computational Fluid Dynamics and Thermal-Hydraulic Modelling Approach for Improving the Thermal Performance of Corrugated Tank Transformers: A Com-

- parative Study. *Energies*, 17(8), 1802. <https://doi.org/10.3390/en17081802>
- Wu, T. and Lei, C. (2015). On numerical modelling of conjugate turbulent natural convection and radiation in a differentially heated cavity, *International Journal of Heat and Mass Transfer*, 91, 454-466, <https://doi.org/10.1016/j.ijheatmasstransfer.2015.07.113>.
- Xiao, J., Zhang, Z., Hao, Y., Zhao, H. and Liu, X. (2024). Method for Measuring Thermal Flow Field Distribution in Oil-Immersed Transformer Using Dynamic Heat Transfer Coefficient, in *IEEE Transactions on Instrumentation and Measurement*, vol. 73, pp. 1-11, Art no. 9002011, doi: 10.1109/TIM.2024.3374310
- Xie, Z., Gu, X. and Shen, Y. (2022). A Machine Learning Study of Predicting Mixing and Segregation Behaviors in a Bidisperse Solid-Liquid Fluidized Bed. *Industrial & Engineering Chemistry Research*, 61, 24, 8551 - 8565, <https://doi.org/10.1021/acs.iecr.2c00071>
- Yang, P., Huang, C., Zhang, Z., Long, T. and Liu, M. (2021). Simulating natural convection with high Rayleigh numbers using the Smoothed Particle Hydrodynamics method, *International Journal of Heat and Mass Transfer*, 166, 120758, <https://doi.org/10.1016/j.ijheatmasstransfer.2020.120758>
- Yang, S.S., Li, D.Y., Kupka, D., Kolonicny, J. and Singh, P. (2024). CFD numerical simulation, verification with experimental results and internal flow analysis of bristle turbines, *Case Studies in Thermal Engineering*, 64, 105436, <https://doi.org/10.1016/j.csite.2024.105436>
- Zhang, Y., Jia, D., Qu, F., Bai, J. and Toropov, V. (2024). A mid-range approximation method assisted by trust region strategy for aerodynamic shape optimization, *Applied Mathematical Modelling*, 132, 607-629, <https://doi.org/10.1016/j.apm.2024.05.005>
- Zhao, S. (2022). Complete-Cooling-Loop Based Transformer Thermal Studies through CFD Simulations and Experimentation, Doctoral dissertation, The University of Manchester. <https://research.manchester.ac.uk/en/studentTheses/complete-cooling-loop-based-transformer-thermal-studies-through-c>
- Zhu, X. (2024). An energy stable SPH method for fluid flow in geological porous media, PhD Thesis, KAUST Research Repository. <https://doi.org/10.25781/KAUST-Q4365>

SAŽETAK

Pregled recentnih istraživanja na području modeliranja spregnutoga prijenosa topline u transformatorima

Točan položaj i iznos najviše temperature materijala izolacije unutar transformatora proračunski se može utvrditi korištenjem računalne dinamike fluida. Međutim, ovaj pristup zahtijeva korištenje znatnih računalnih resursa. U ovome radu napravljen je pregled nekih nedavnih istraživanja na ovu temu fokusirajući se pritom ponajprije na istraživanja u kojima je u obzir uzeta prostorna distribucija zavisnih varijabli u problemu spregnutoga prijenosa topline. Određivanje najviše temperature izolacije unutar transformatora još uvijek je otvoreno pitanje. Nadalje, napravljen je osvrt na fundamentalne aspekte vezane uz prijenos topline unutar transformatora te su predloženi neki novi pristupi za proračun prijenosa topline transformatora: ugađena hidrodinamika čestica, dvofluidni model i primjena računalnoga programa T-Flows. Tako se ugađena hidrodinamika čestica pokazala prikladnim pristupom jer je tradicionalno vezana uz laminarno strujenje kakvo se pojavljuje unutar transformatora; dvofluidni model znatno šteti računalne resurse, dok je primjena računalnoga programa T-Flows besplatna, a izrazito snažna za proračun spregnutoga prijenosa topline i paralelno računanje. Provedenim se istraživanjem definiraju smjernice za daljnji rad na području numeričkoga modeliranja prijenosa topline unutar transformatora.

Ključne riječi:

transformator, spregnuti prijenos topline, računalna dinamika fluida, najviša temperatura izolacije

Author's contribution

Alen Cukrov (senior assistant, PhD, mechanical engineering) contributed with the paper conceptualization, investigation, writing - original draft and visualization. **Ivanka Boras** (full professor tenure, PhD, mechanical engineering) contributed with supervision and funding acquisition. **Martin Jurina** (MSc of Mechanical Engineering, thermodynamics development engineer at Končar D&ST) contributed to project administration. **Dino Kovačević** (MSc of Mechanical Engineering, thermal engineer at Končar D&ST) contributed to review and editing. **Branimir Čučić** (PhD, Department Head at Končar D&ST) contributed to conceptualization and supervision of the project. All authors have read and agreed to the published version of the manuscript.