Marek LIS[1], Michał KOBIERSKI[2], Marek CHMIEL[3]

Politechnika Częstochowska Wydział Elektryczny
ORCID: 1.00000-0001-5191-8303; 3.0000-0001-8662-7127

Induction heating as a process of converting eddy current losses into thermal energy in the Ansys Coupling software environment

Abstract. The article presents the use of "Ansys" software for multi-environment simulation. The calculations of the impact of the heat source (eddy current losses) and the distribution of heat energy in heat-conducting environments (steel, air) were combined. The phenomenon of heat dissipation in the vicinity of the heated steel element (heating of the air and thus the possibility of starting a fire) was taken into account. Calculations of the temperature of the heated elements of the device were made. The temperature state of the considered elements was linked with the linear movement of the heating objects (sliding movement of the heated element - dynamics of heating).

*Streszczenie. W artykule przedstawiono wykorzystanie oprogramowania „Ansys" do symulacji wielośrodowiskowej. Połączono obliczenia wpływu źródła ciepła (straty wiroprądowe) i rozkładu energii cieplnej w środowiskach przewodzących ciepło (stal, powietrze). Uwzględniono zjawisko wydzielania ciepła w sąsiedztwie nagrzewanego elementu stalowego (nagrzewanie się powietrza i tym samym możliwość wzniecenia pożaru). Dokonano obliczeń temperatury nagrzewanych elementów urządzenia. Stan temperatury rozpatrywanych elementów powiązano z ruchem liniowym obiektów grzejnych (ruch liniowy nagrzewanego elementu – dynamika nagrzewania). (**Nagrzewanie indukcyjne jako proces przemiany strat wiroprądowych w energię cieplną w środowisku programowym Ansys Coupling**)*

Keywords: symulation, Ansys, coupling, program
Słowa kluczowe: symulacja, Ansys, sprzęganie, program.

Introduction

The Ansys environment is a universal tool that allows for a very wide range of searching for real technical solutions in the virtual space. The savings of linking virtual experiences with their reflection in reality cannot be overestimated. The Ansys environment offers a very wide range of possibilities for computer simulation linked to reality. Simulations whose physical realities and mathematical descriptions are often very distant require the independent use of many specialized programs. As an example, calculations of electrical losses related to the flow of thermal energy in the same real object (steel bar, induction heating, resistive losses) and the surroundings can be given. This phenomenon is well known in the construction of electrical devices. Under the influence of eddy current losses, the components of the device heat up (induction heating, hardened object), the heat is displaced and dispersed in the surroundings (possibility of initiating a fire).

The article presents the use of Ansys programs to analyze the system shown in Figure 1.

Description of the problem

The simulation presented in the article covers the issue depicted in Figure 1. The steel bar, marked in the drawing, is surrounded by a copper coil - exciter, in which a current of changed frequency flows. The frequency of the current in the coil is considered as one of the system variables, because changing the supply frequency affects the amount of eddy current losses. Resistive losses affect the amount of energy that will be converted into heat in the object. In the virtual experiment considered in this work, the authors compared the impact of changing the frequency in the range of 60Hz to 3kHz. During the simulation, the area and interior of the bar located inside the coil - exciter were taken into account. This is an area corresponding to the approximately 0.5 of the length of the bar. The space was selected in such a way that it was possible to simultaneously observe the flow of thermal energy in the volume of the tested bar. Due to the principle of transferring and generating thermal energy, this phenomenon is known as induction heating. Recently observed changes in the natural environment inspire the scientific community to conduct research in this direction. Transferring energy

through a magnetic field allows not only the study of phenomena related to electrical losses in operating electrical devices, but also heating devices (induction hardening), devices such as induction cookers or induction water heating devices in households. Photovoltaic sources of electricity, due to the relatively low voltage of such sources, allow the use of solar energy to charge electric car batteries. By wirelessly transmitting electricity directly to the battery, there is no need to connect their to the network. The authors present one of the methods of transmitting electrical energy, which is converted into thermal energy and used at the destination.

Calculation method

All calculations were performed based on the Ansys Workbench 2021/2023 environment. The part used for simulation was made based on the Maxwell program in accordance with Maxwell's Finite Elements method (3). It requires zero boundary conditions established around the space surrounding the considered object. The space is divided into approximately 300,000 finite elements in the form of triangles, quadrangles on the surfaces of the object in question, and approximately 400,000 tetrahedrons of various sizes making up the interior of the device. Programs of this type require very high-performance computers and large memory areas. The number of finite elements is important due to the precision and speed of calculations and should be additionally concentrated in areas important for accuracy. Particular attention should be paid to areas such as air gaps, sharp edges of pole shoes or magnetic circuits. Rounded elements (with a circular cross-section, curved edges) also pose an additional difficulty for calculations due to the shape of triangles and tetrahedrons, with the help of which these surfaces must be represented. When drawing a device, you sometimes need to present shapes using polyhedra as in Figure 1.

This shape greatly facilitates the design of the finite element mesh, while reducing the amount of erroneous calculations. The Ansys program distinguishes between elements that move and remain in a rigid system. This is due to the need to arrange a finite element mesh every time the relative position of the machine parts changes.
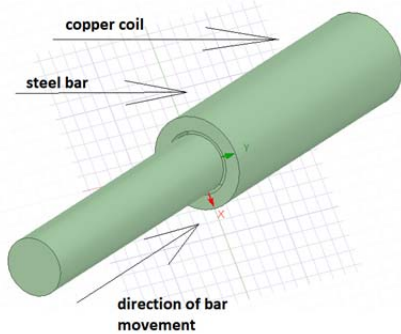
Fig. 1. Description of the phenomenon analyzed in the article, implemented based on the circular structure.

After calculating the field intensity distribution of the B[T] vector (1)

$$(1) \qquad \vec{B} = \mu_r * \mu_0 \vec{H}$$

where

$$(2) \qquad \oint_L \vec{H} d\vec{l} = I + \frac{d\Phi_D}{dt}$$

$d\Phi_D$ is the electric induction flux [C = As] and I in formula (2) is the current density (in 3D space), it is possible to calculate the current values of individual finite elements in subsequent time instants dt.

Based on the calculated values, knowing the values of current I and resistance R (due to the properties of the material, in this case the resistivity ρ of steel), the ohmic losses can be calculated in turn (Chart 1):

$$(3) \qquad P = I^2 * R$$

where the material resistance is calculated based on the resistivity of the analyzed material (in 3D space).

$$(4) \qquad R = \frac{\rho * l}{s}$$

The parameters ρ, l and s (of the cross-sectional area of the finite element with the properties of an electrical conductor) result from the dimensions of the finite elements assumed at the stage of designing the device geometry and the 3D mesh of the computational space. After calculating the amount of energy loss per volume of the analyzed material, its temperature can be calculated. Having information about the size of the current vectors, it is also possible to calculate the mechanical parameters related to the analyzed device, where F is the calculated force acting on the finite element according to formula (5)

$$(5) \qquad F = \int \left[\frac{1}{\mu_0} B(B.\vec{n}) - \frac{1}{2\mu_0} B^2 .\vec{n} \right] dC$$

The forces pulling the steel core into the coil, calculated according to formula (5), act in the direction normal to the surface in accordance with the unit vector $\vec{n}$. It should be noted the direction of the unit vector, which determines the direction of the force. The integral over the volume of the material gives information about total force acting on the mechanical element. Using formula (6), the torsional moments acting on the analyzed steel bar can be determined.

$$(6) \qquad T = r * F$$

Quantities calculated based on formulas (1, 2, 3, 4, 5) and (6) along with the B[T] induction values calculated in the simulation with a unit vector n indicating the direction of the acting force, induction, current or torque, taking into account the magnitude constant magnetic permeability for vacuum $\mu_0 = 4\pi10^{-7}$ [H/m] are true provided that the appropriate proportion, greater than 1/1000, is maintained between the magnetic permeability of the vacuum and the permeability of the construction material of the analyzed device. In the case of analyzing a device constructed of steel in an air and vacuum environment (permeability of steel, air, air gap), these conditions are met.

Calculation results

Due to the possibility of access to the PIONIER Computing Platform and the possibility of using large memory areas and fast processors, a design was used without the need to use multi-plane shapes. Figure 2 shows the design analyzed in this paper. Attention should be paid to the structure based on smooth curves, and the number of finite elements used for this type of analysis allowed for calculations with the number of erroneous elements below 1%.
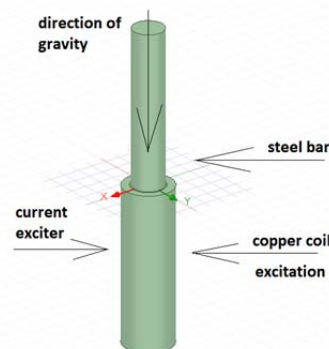

Fig. 2. Description of the phenomenon analyzed in the article, based on the structure of smooth curves

Figure 2 shows the direction of gravity. This information is necessary due to the effect of the hot element on the air. Hot air flowing against gravity can cause a fire if not properly controlled. The hot air flow is shown in Fig. 5.

According to the assumptions for the preliminary design, the device consists of a copper coil powered by alternating current at frequency f. The field generated by the coil causes the appearance of eddy currents, which flow on the surface of the bar (current displacement phenomenon) and causes it to heat up. The Ansys Maxwell program only allows you to calculate losses related to the resistance of inductively heated material (Ohmic Losses). In this way, it is only possible to estimate the losses (heating) based on the energy density of the losses (the unit is W/kg).

In the case of the first calculations (Table 1), the losses amount to e.g. 2.14 W/kg of steel. Please note that Ansys converts values in SI units. Table 1 shows the losses in units more frequently used in engineering practice. The amount of losses was converted into W/kg. Taking into account the weight of the bar (approx. 2 kg) and ohmic losses at a frequency of 3 kHz, it can be assumed that this amount of steel (approx. 2 kg) will be heated in a few seconds. Calculating the heating time requires additional calculations and the authors will do this in the next stages they intend to perform. The results of calculations of the amount of losses inside the steel bar at a frequency of 60 Hz are displayed on the cross-section of the bar, i.e. 383 kW/m^3, which corresponds to approximately 2 W/kg at a steel density of 7.8 g/cm^3. The size results from the scale of the chart and the maximum values presented on the

chart (red color on the chart in small areas). An example diagram of the distribution of ohmic losses in the bar is shown in Figure 3.
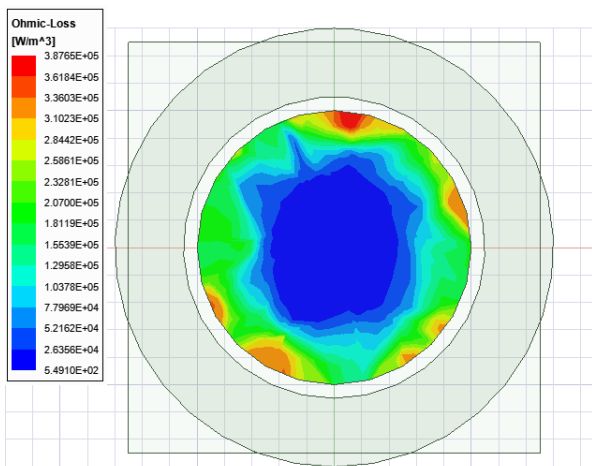

Fig. 3. Ohmic losses in the bar: cross-section of the bar at ½ of the coil (see Fig. 2)

Taking the above into account, the energy released in the heated bar should be calculated according to the formula:

$$(7) \qquad E = \oint OmicLoss \cdot dV$$

This type of procedure allows you to calculate the actual value of the energy transferred by the coil to the steel bar material by using a specialized calculator that can be found in the Maxwell program. Additionally, this function can be performed by passing the calculations to a Maxwell-linked program called IcePack. This is an Ansys environment program that can calculate the steady-state temperature of the bar through proper coupling. It should be noted that the results can only be interpreted as "steady state".

Fig. 4 Scheme of "coupling" in the process of calculating the bar temperature (two separate programs calculating the bar temperature)

Table 1 Simulation results

No.	I_{field} [kA]	f_{field} [Hz]	B [T]	Loss [W/kg]
1	0.8	60	0,29	2,14
2	0.8	300	0,68	25,10
3	0.8kA	1000	1,12	159,24
4	0.8kA	2000	1,30	402,55
5	0.8kA	3000	1,43	1566,88

After transferring the calculation data several times (the number of transfers allows you to increase the accuracy of the simulated temperature), you can obtain information about the precisely calculated bar temperature. Additionally, the program allows for the assessment and measurement of temperature distribution in the volume of the bar (e.g. along the axis of the bar) based on the image of the temperature distribution anywhere, both on the surface and throughout the entire volume of the bar.

To obtain the above results (Table 1), it was necessary to use programs' coupling in 4 cycles (this means stabilization after the third approximation). Examples of the results of the power demand needed to heat the bar to the temperature shown in the temperature distribution charts

are given in Figure 5, and the results of the power demand calculations in subsequent calculation cycles are given in Table 2.

Table 2. Power demand to heat the bar

Cycle no.	f_{field} [Hz]	P_{bar_loss} [W]	I_{field} [kA]
1	60	197,5	0,8
2	300	271,4	0,8
3	1000	2.202	0,8

It is clear from Table 2 that subsequent cycles approach the steady-state value after the cycle corrections are introduced. After the second cycle, the power transferred to heat the bar is less than 300 W.

The temperature of 693°C does not result from the average bar temperature but from the total temperature of the bar and coil. The program took into account the heating of a coil through which a current of 5 kA flows. Additionally, you can observe the flow of hot air, which expands in the program in the proportion of 0.0001 per 1°C. This causes hot air to rise in the area used as boundary conditions in the program. The ambient temperature outside the calculation area is 22°C, which can be observed by the temperature distribution (blue areas) in the space above the simulated device. In the future, the authors intend to study heat flows in electrical devices, their cooling processes and the risks associated with the occurrence of uncontrolled or unforeseen by designers phenomena and situations related to uncontrolled heating of the device. Simulations in 3D space facilitate this type of research without much expense.
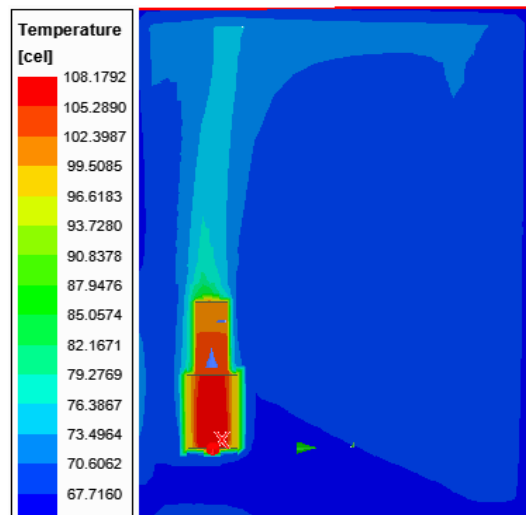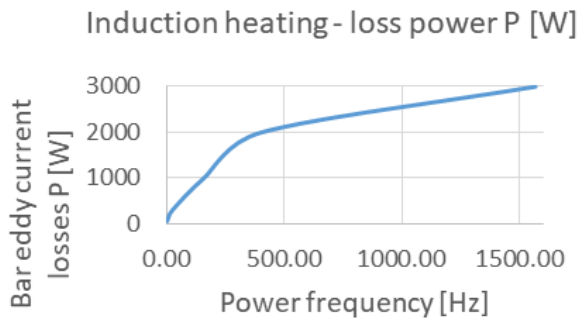

Fig. 5. Temperature distribution in the calculation area. Heated air flow is shown.
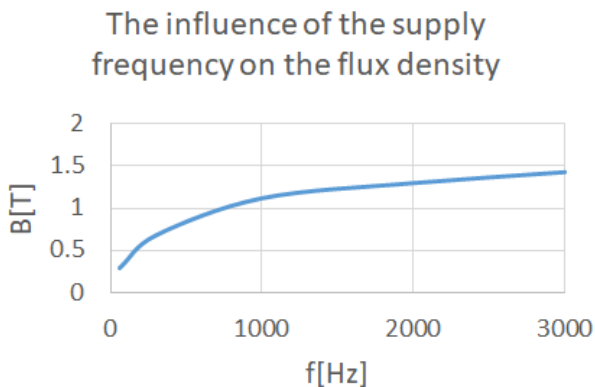
Discussion

The calculations additionally took into account the dynamic change in copper resistance (the dependence of the change in copper resistance on temperature) and the dependence of the change in air volume on its temperature. The effect of this type of change can be observed by causing a change in the direction of gravity in the simulation and the effects of this type of behavior when changing the data of the virtual experiment.

Changing the direction of gravity may allow for the simulation of temperature transfer conditions and the causes of fire ignition. This also allows for the proper arrangement of heated elements to avoid the danger of fire initiation. The simulation was performed for several parameters with their changing assumed values. Graphs of these relationships are presented as Graph 1 and Graph 2.

Induction heating - loss power P [W]

Graph 1. Dependence of losses in the heated bar on the power supply frequency

The influence of the supply frequency on the flux density

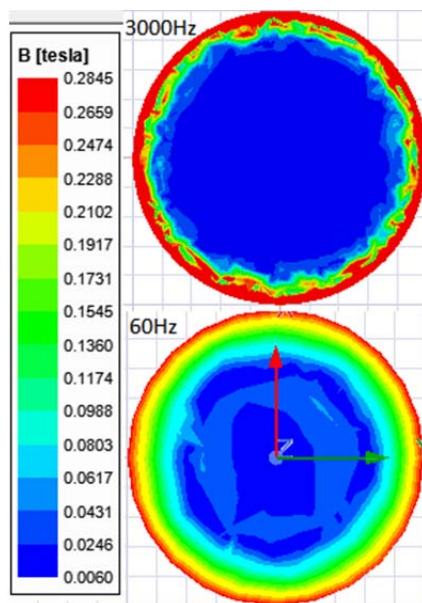Graph 2. Dependence of flux density on power frequency

Fig. 6 Comparison of the B[T] flux density distribution in the bar cross-section at different current frequencies of the exciter coil.

A significant increase in ohmic losses can be seen with increasing supply frequency. Graph 2 shows the relationship between the maximum value of induction B[T] when changing frequency. The saturation of the material with a simultaneous increase in the power frequency is clearly visible. Similar observations related to the skin effect can be observed in the distribution of magnetic induction B inside the bar (Fig. 6).

Figure 6 shows a cross-section of an inductively heated bar when changing the frequency of the supply voltage as described for 60 Hz and 3000 Hz. The drawings show a reduction in the area of influence of the magnetic field. At a frequency of 60 Hz, the area has a larger field impact area and at a frequency of 3000 Hz, the area has a smaller field impact area but at the same time an increased flux density value.

Conlusions

The authors present the possibility of simulating phenomena related to the occurrence of eddy currents using the Ansys environment. Software development allows for the connection of electrical and thermal phenomena. Due to the specialization of individual program blocks, it is necessary to use couplings between the blocks and data calculated by individual programs. The new system in the extensive Ansys environment allows for this type of coupling using a convenient user interface. The "Coupling" system allows you to combine the results of the Maxwell program used to calculate electromagnetic fields with the "Mechanical" or "IcePack" programs used to calculate thermal phenomena. This way, the designer can observe thermal and electromagnetic phenomena simultaneously. In the example presented, thermal phenomena related to electromagnetic phenomena can be observed. The connection between ohmic losses and eddy current losses makes us aware of the reason for increasing the losses and at the same time increasing the intensity of heating the material. However, there is an important detail to note about the example. The method used in both Maxwell and IcePack does not introduce the concept of time. The describing equations do not have a time parameter in their description. This is a "Stedy State" type. To obtain the dependence of the heating processes on time, a "Transient" analysis should be used. The Ansys environment has this capability. The authors intend to use this procedure in further research. This will allow the heating processes to be correlated with real time (e.g. field operation time with the final temperature of the heated object) and enable simulation in the virtual space to cooperate with the design work of creating real devices.

Authors: dr hab. inż. Marek Lis, Politechnika Częstochowska, Wydział Elektryczny, marek.lis@pcz.pl., mgr inż. Michał Kobierski, Politechnika Częstochowska, Szkoła Doktorska, fireman666@o2.pl, dr inż. Marek Chmiel, Politechnika Częstochowska, Wydział Elektryczny, marek.chmiel@pcz.pl.

REFERENCES

[1] *Ansys, System coupling Tutorials, Dokumentacja Ansys 2023*
[2] Halmann D., Modelowanie Układów Elektromagnetycznych w środowisku Ansys (Ansys Electronics Desktop), Uniwersytet Morski w Gdyni ISBN 978-83-7421-383-7
[3] Maxwell 15.0 application User's Guide
[4] Ansys 14.0 application User's Guide
[5] .WorkBench 14 application User's Guide
[6] Wang Zai-Zhou Zhang Chengning Song Qiang Fan Jin-xin, Zhang Chun-xiang; Research on noise source identification of traction motor system for electric buses based on sound intensity, Sch. of Mech. & Vehicular Eng., Beijing Inst. Of Technol., Beijing , Vehicle Power and Propulsion Conference, 2008. VPPC '08. IEEE