

FEM SIMULATION INDUCTION HEATING PROCESS

Josef HODEK

COMTES FHT a.s. Dobruška, Czech Republic, EU, josef.hodek@comtesfht.cz

Abstract

Induction heating is today wide used technology in industry in huge range of fields. Solution of this process contains both solution of the electromagnetic field and solution of the thermal field. In the past were knowledge and experiences obtained mainly from experiments but at the present time is possible to study induction heating phenomena using modern numerical method especially the finite element method [1]. The numerical solution helps to understand induction heating more complex and points out critical issues. Big advantage is use the numerical solution for induction heating coil design. This approach can avoid of long-lasting and expensive experiments with different variants.

This paper deals with FEM simulations of induction heating. There are electromagnetic and thermal field taken into the account. The material properties are obtained on the base of material chemical composition. The numerical model considers thermal dependant material properties. Different coil design is considered and variants are compared.

Keywords: finite element method, induction heating, numerical simulation, Deform

1. INTRODUCTION

Induction heating is wide spread process in the metal heat treatment. In comparison to traditional heating process has induction heating several advantages like heat speed, heat targeting, flexibility and better automation. Those advantages make induction heating more efficient more environment friendly process.

In case of heating of complex workpiece there is no evident way how to design induction coil. Induction heating process - collection thermal and electromagnetic phenomena - is possible to solve by help modern numerical method (the most used Finite Element Method). With the FEM based software DEFORM3D [2] is carried out to determination of induction coil design from given possibilities.

2. THEORETICAL BACKGROUND

Electromagnetic field generated by the inductor induces eddy currents in electrically conductive materials. These eddy currents cause heat to develop in the material. The heating process can be controlled through the inductor shape. An FEM model of the induction heating process combines the computations of electromagnetic and temperature fields. Such a model allows this phenomenon to be analysed to a great depth. The present model couples the calculations of electromagnetic quantities and temperatures, i.e. the material property data are updated continuously on the basis of the current temperature.

The temperature field is described using the Fourier equation:

$$c\rho\dot{T} = \text{div}(\lambda \text{grad } T) + w$$

Where w stands for eddy current losses

The electromagnetic field [3] is described through equations for the magnetic vector and scalar potentials:

$$\text{rot}\left(\frac{1}{\mu}\text{rot } A\right) = J - \sigma(\dot{A} + \nabla\varphi)$$

$$\nabla\sigma(\dot{A} + \nabla\varphi) = 0$$

Discretization of the model of the area of interest creates model elements and transforms partial differential equations into a system of algebraic equations that can be solved using conventional methods. Adding boundary conditions provides an unambiguous mathematical definition of the problem.

General FEM model (see **Fig. 1**) is solved in DEFORM 3D simulation engine that performs the numerical calculation.

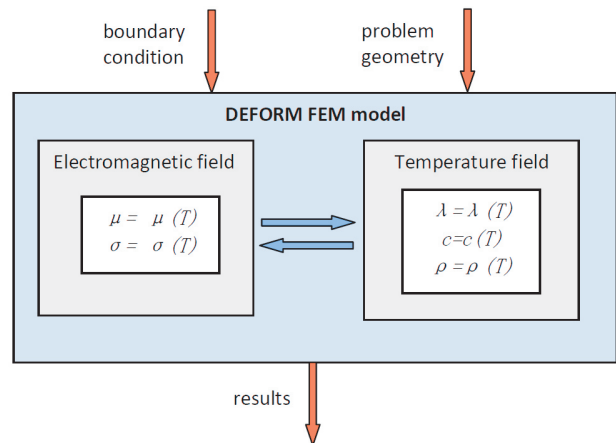


Fig. 1 FEM model

3. SIMULATION MODEL

Heat treated workpiece had to be heated just in the upper part see **Fig. 2** to obtain desired properties. The most important requirement was to reach sufficient uniform temperature distribution in the heated area. The workpiece weight was about 150kg.

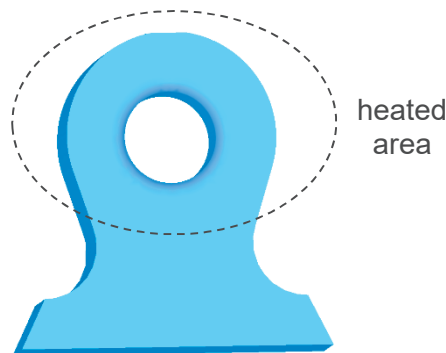


Fig. 2 Treated workpiece with marked heated area

Thanks to plane symmetry, only one half of the workpiece could be used for computation. The heat transfer coefficient between the workpiece and environment was set to 200W/m²K. Temperature-dependent materials characteristics of the workpiece were obtained from calculation in JMatPro on the basis of the material's chemical composition. Inducted thermal power was same for both alternatives and frequency of the current was set to 1kHz. Heat treating temperature was 620 °C. Finite element mesh had about 25000 elements.

According to induction coil construction were chosen two coils (see **Fig. 3**, **Fig. 4**) that were examined and compared. The first coil (see **Fig. 3**) is simple cylinder and the second coil (see **Fig. 4**) is shaped according the workpiece headed body. Both figures have marked the coil current direction.

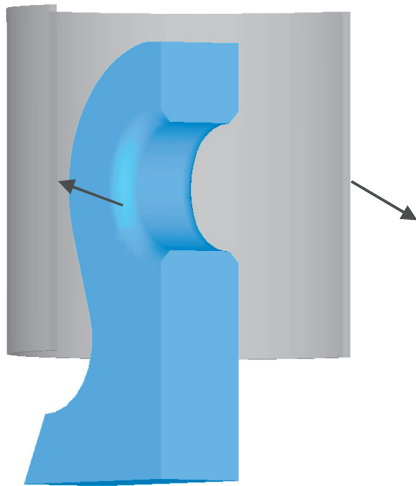


Fig. 3 Coil 1 (current direction marked)

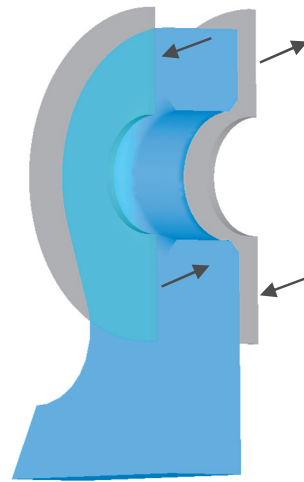


Fig. 4 Coil 2 (current direction marked)

4. RESULTS

The temperature distribution for both coils were computed. The significant difference between heating pattern was found. The first coil heated the workpiece very unequal (see **Fig. 5**). The maximal body temperature was reached in palces close to inductor. The second coil heating patern (see **Fig. 6**) had better temperature distribution.

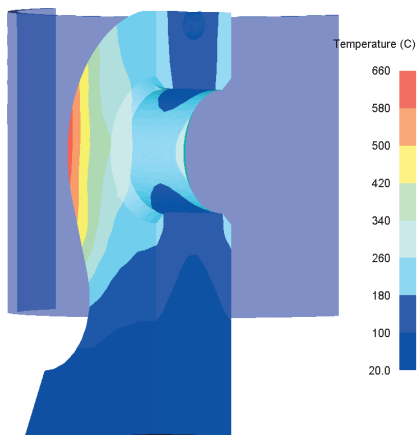


Fig. 5 Coil 1 temperature distribution

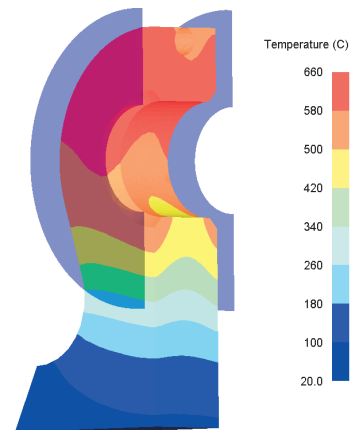


Fig. 6 Coil 2 temperature distribution

CONCLUSION

The above presented results show that how can be used FEM simulation for induction coil design. Induction heating is combination electromagnetic and thermal phenomena and DEFORM 3D software was used for solution. The boundary conditions for both electromagnetic and thermal phenomena were applied. Two induction coils were computed and the results were compared.

The second induction coil had better temperature distribution and this variant suited more to the workpiece treatment. The results weren't evident before the solution and the FEM simulation showed the way of the coil design without experimental testing.

ACKNOWLEDGEMENTS

This work was supported by the grant MSM/ED2.1.00/03.0077

REFERENCES

- [1] ŠUCHMAN, P., HODEK, J., Optimization of Induction Hardening Parameters through FEM Simulation, 20th IFHTSE Congress, October 22-25 2012, Beijing
- [2] DEFORM[™], Simulation of Induction Hardening, DEFORM Application #502
- [3] LANGER E., Teorie indukčního a dielektrického tepla, ACADEMICA Praha 1979