Evaluation of Electromagnetism capabilities of LS-DYNA®: alternative heating processes

Edith Grippon[1], Timothy Senart, Vincent Lapoujade[1],

[1]DynaS+, 5 Avenue Didier Daurat, 31400 TOULOUSE, France

1 Introduction

Hot stamping is one way to produce automotive steel parts of complex shape with very high mechanical characteristics in selected areas. In this process heat plays an essential role. To reduce investment costs and for more production flexibility during the initial heating step, automotive manufacturers look into using alternative heating techniques.
The main subject presented in the paper is a way to reduce the heating time and to offer more flexibility when increasing the temperature of the automotive steel blanks, based on Joule effect heating. Two techniques will be explained, inductive and resistive heating, focusing on numerical aspect in order to help manufacturers to reduce the time of setting up such processes. Models using Lagrangian, Thermal and EM solvers will be presented with a comparison to experimental data.

1.1 General introduction

Whatever the intended applications (aeronautics, automotive, naval, etc.), hot forming processes require preheating. The controlled deformability of hot steel allows to establish complex part geometries without adding reinforcement, welding or assembly point. However, two issues arise from this type of methodology: the significant heating time and the wear of the shaping tools.
To reduce the energy resources necessary for the heaters and implementation time, large steel companies are investigating alternative heating techniques. One of them, proposed by ArcelorMittal Global R&D Montataire, is the use of electrical power to create a magnetic field that will heat conductive parts by induced current. The optimization of this method requires numerical and experimental studies. Indeed, the integration of digital simulation upstream of the design process phase helps limits the loss of raw material (manufacturing tools, tests on plates) and energy (immobilization of production machines). It is a very useful way of analyzing and understanding the heating and deformation mechanisms of the steel sheets during the various operations.
Following a brief review on the LS-DYNA EM solver, in a first part, electromagnetic induction heating study will be presented. In the second part, an industrial study corresponding to a resistive heater will be presented.

1.2 The EM solver for induced and resistive heating applications

The default LS-DYNA EM solver is the Eddy current solver and allows to solve the induction-diffusion effects over time with any current/tension input shape including periodic oscillatory behavior (sinusoidal current). The electromagnetic fields are solved using a Finite Element Method (FEM) for the conductors coupled with a Boundary Element Method (BEM) for the surrounding air/insulators. Both methods use elements based on discrete differential forms for improved accuracy [1]. Contrast to conventional methods, the advantage of the Boundary Element Method used in LS-DYNA comes in the ability to simulate the interaction between different conductor parts without meshing the air volume. In Electromagnetic metal forming context, the fact of not remeshing air at each time increment is particularly interesting.

1.2.1 Thermal coupling

Both the thermal and the electromagnetism solver run implicitly. At each electromagnetic time step, the electromagnetism solver will communicate the extra Joule heating power term and the thermal solver will communicate the temperature. Several equations of state are implemented in the electromagnetism solver that permit to define how the conductivity is evolving as a function of the temperature.
In this paper, inductive and resistive heating calculations do not take into account the strains and stresses in materials. The mechanical solver is not active. The coupling is made only between EM and the thermal solver. The electrical conductivity of the material must be specified. The influence of the

capacity and thermal conductivity of materials, of the coefficients of convection and radiation to the thermal boundary conditions are taken into account.

1.2.2 The inductive heating solver

Induction heating is the process of heating an electrically conducting object (usually a metal) by electromagnetic induction (usually with a coil), where eddy currents are generated within the metal and resistance leads to Joule heating of the metal. The solver works in the time domain and not in the frequency domain, in order to easily take into account coil/workpiece motion as well as the time evolution of the EM parameters. Therefore, in order to solve an eddy current problem, an EM time step compatible with the frequency is needed.
The induction heating solver was introduced in order to solve the computer cost issue arising when high frequency currents (very small time steps) are combined with long simulations runs (typically, a AC current with a frequency ranging from kHz to MHz and a total time for the process around a few seconds) [2]. In this condition, the following assumption is done (figure 1):

- a full eddy-current problem is solved on one full period using a "micro" EM time step. An average of the EM fields and joule heating energy during this period are computed.
- It is then assumed that the properties of the material (and mostly the electrical conductivity which drives the flow of the current and the joule heating) do not significantly change over a certain number of oscillation periods delimited by a "macro" time step. These properties depending mostly on the temperature, the assumption can therefore be considered accurate as long as the temperature does not change too much. During these periods, no EM computation is done, only the averaged joule heating power is given to the thermal solver.
- But, as the temperature changes, and thus the electrical conductivity, the EM fields need to be updated accordingly, so another full eddy current resolution is computed in the "macro" time step.

Contrary to the FEM matrices, the BEM matrices are full dense and cannot be stored as dense arrays since the memory requirement would grow up very quickly with the size of the system (times consuming) [3], [4] and [5]. Therefore, does not reverse the BEM matrix represent a considerable win of run time.

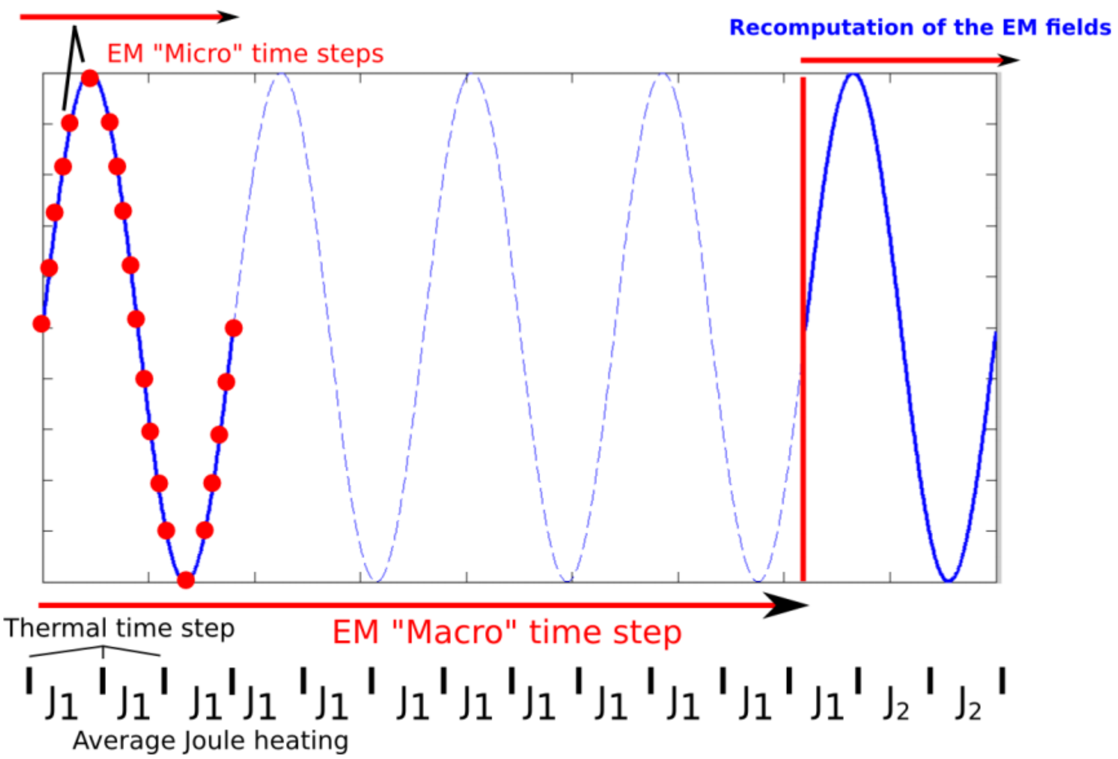


Fig. 1: Inductive heating solver [6]. The user can specify the number of the "micro" EM step in a quarter period. The user can define the "macro" time step when the EM fields are recomputed on a new quarter period.

1.2.3 The resistive heating solver

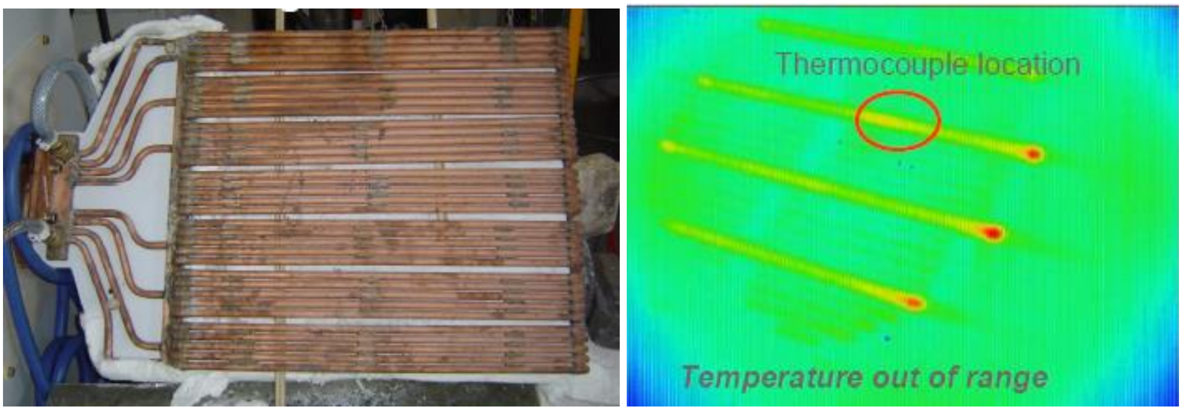
The resistive heating solver is a simplified version of the eddy current model where only resistive and no inductive effects are computed. The vector potential is equal to zero all over and only the scalar potential is kept. Therefore only circuits that impose the tension can be used for this solver. This model is for very slow rising currents in a piece connected to a generator, where the diffusion and inductive effects can be considered as infinitely fast. The joule heating due to the current is still taken into account but no mechanical force is generated. Very large time steps can be used and no BEM system is needed, this makes this solver much faster than the full eddy current model.

2 Inductive heating

The aim is to set up an induction heating model compatible with industrial constraints in terms of computing time and presenting a realistic temperature evolution.

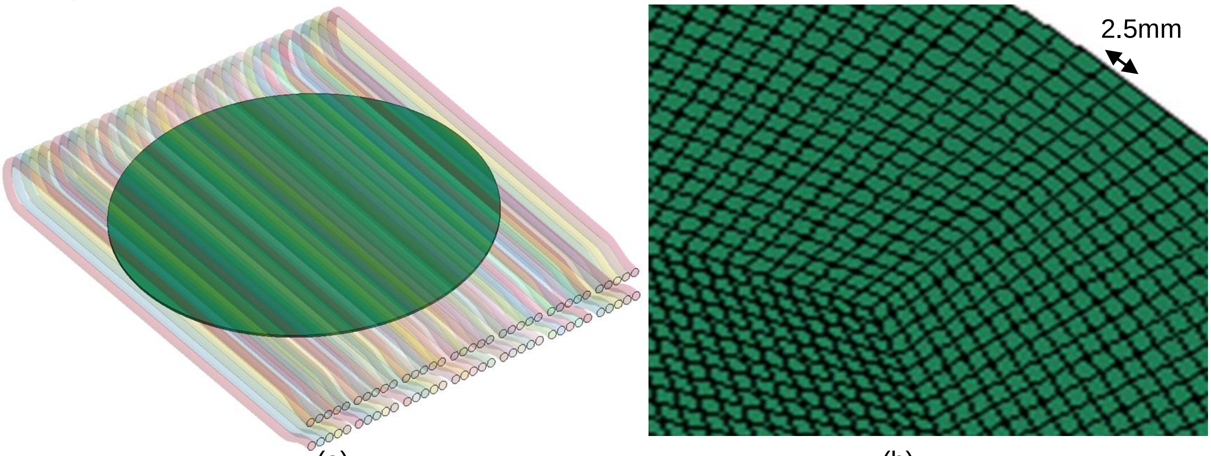
2.1 Presentation of the case study

The experimental heating system set up by ArcelorMittal Global R&D Montataire has 35 copper coils (Figure 2 (a) and Figure 3 (a)). This system is powered by a generator delivering an electrical signal. The electric current in the coils creates an electromagnetic field (induced current) for heating by Joule effect a conductive circular plate. This steel plate has a thickness of 2.5mm and a diameter of 400mm. The heating time is 10s, local heating phenomena are observed experimentally (Figure 2 (b)). These phenomena have oriented the choice of the mesh size of the steel plate (Figure 3 (b)).



(a) (b)

Fig. 2: (a) experimental system for inductive heating of a circular steel plate and (b) highlighting heating location



(a) (b)

Fig. 3: (a) modeling of the experimental device for inductive heating of a circular steel plate and (b) mesh of the steel plate

2.2 Model optimization

The elements used for the plate are volume elements with full integration. The materials used for the mechanical part are "rigid body" that reduce the computation time. In addition, the coils are supplied with current to avoid issues related to charges accumulations.

The computation time of BEM matrices are directly related to the number of elements. A sensitivity analysis of the mesh size in coil led to optimize the computation time (Figure 4). Modeling results showed that the use of coils with 12 elements reduces the computation time by 6 and causes an error of 0.3% in terms of the maximum temperature reached in the plate. Therefore, this coil mesh is sufficient for our case. In a further study, where the radiation phenomena are not neglected, a more fine mesh will be consider for better results.

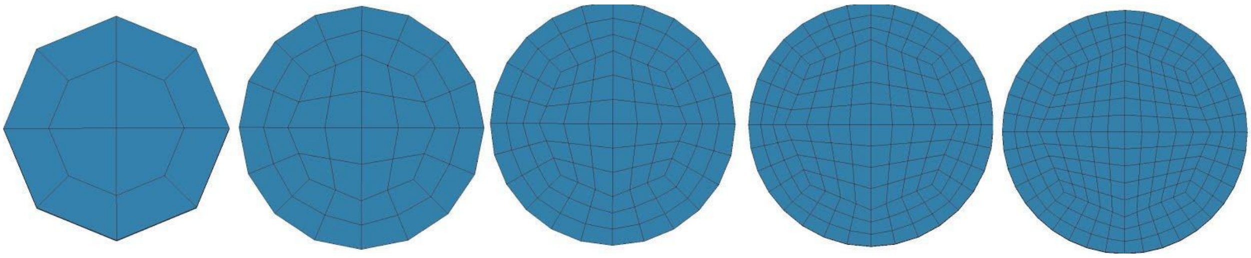


Fig. 4: different mesh sizes for the coils section ranging from 12 to 220 elements

In this cases there is no conductor motion, after the first initial calculation no further BEM calculation is done over the macro time step and the Joule heating is simply added to the thermal solver at each thermal time step. Given the number of interaction between the 35 coils and the plate, the calculation of the BEM is the longest stage in computing time. The fact of not update the BEM in each EM time step significantly reduces the computation time and make this model compatible with industry rates.

The FEM matrix is updated regularly to take into account the changes in material properties (electrical conductivity) due to temperature variations. Figure 5 shows the effect of the update frequency of the FEM matrix on temperature changes in the plate. This difference is explained by the fact that the electrical conductivity of the steel has a growing profile in the temperature interval considered. Thus, higher the temperature increases, the current quantity increases and the heat reached by Joule effect also. In terms of computing time, the update of the FEM matrix changes the model (a) from 40 min to 1h25 for the model (b) of Figure 5.

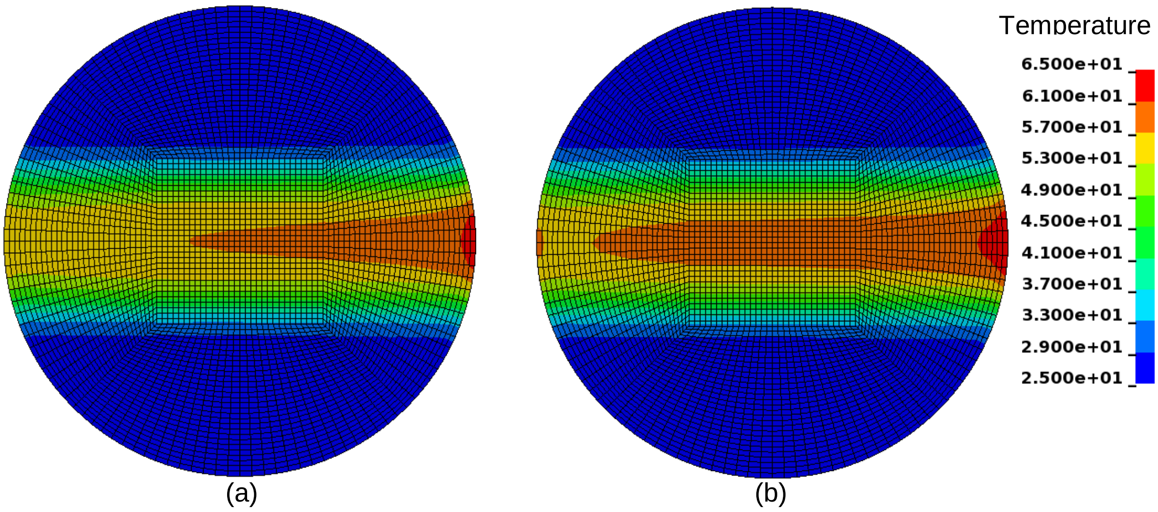


Fig. 5: influence of the frequency of the FEM matrix update for a plate with 10 coils (a) a single update of FEM matrix and (b) 5 updates of the FEM matrix

After optimization of modeling parameters, the entire system with 35 coils (Figure 6) can be modeled over several tens of seconds of study for a computation time of some hours only (for study time of 10s, the computation time is 1h30 on a 16-core machine).

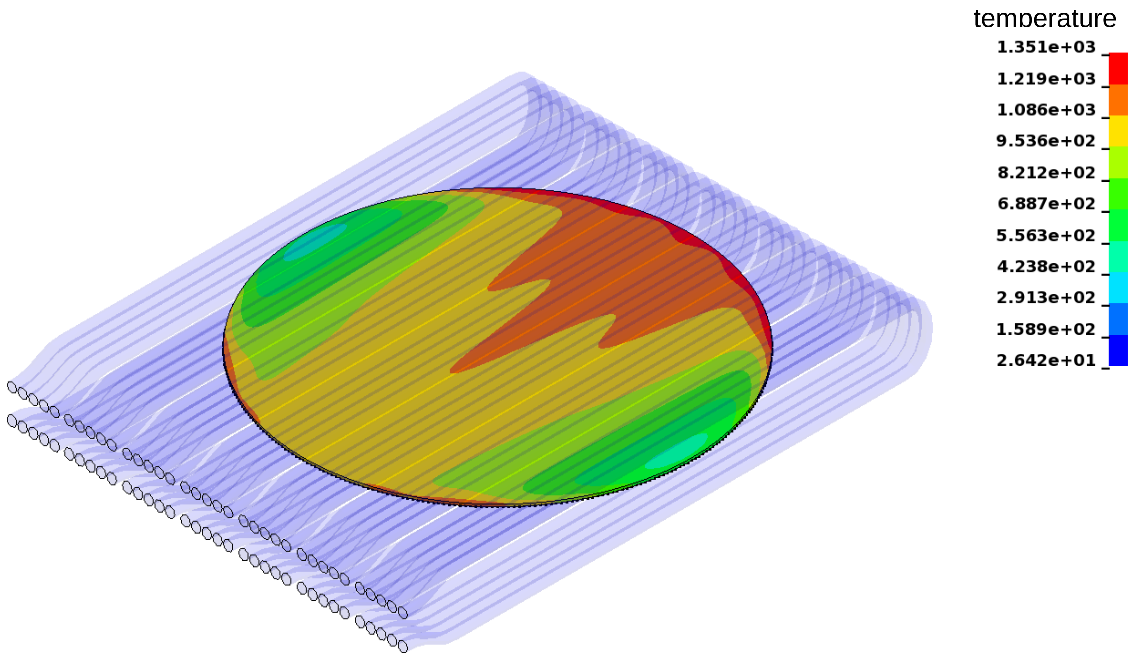
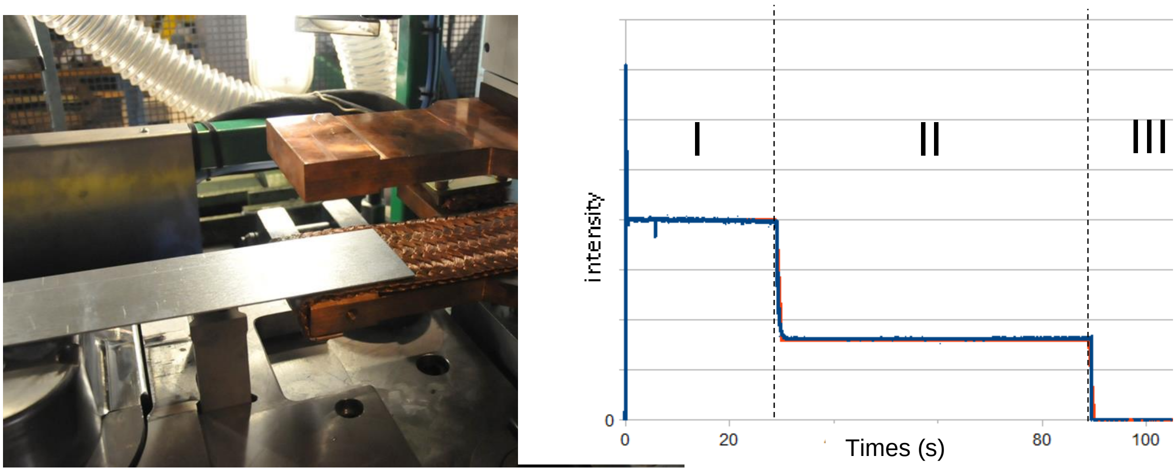


Fig. 5: final model of inductive heating representing the temperature profile at 2.5 s

3 Resistive heating

3.1 Presentation of the case study

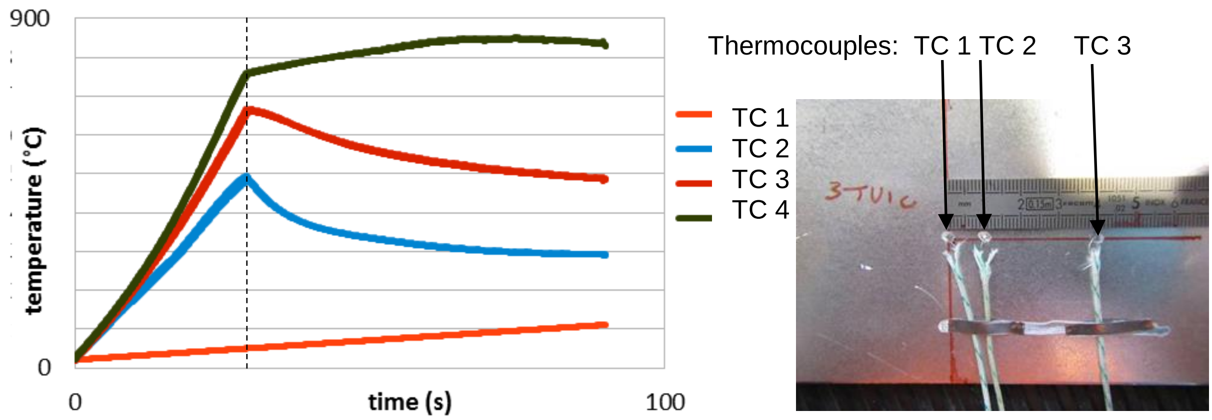
Another proposed technique for the heat of steel plate is the resistive heating. This technique requires a contact between the electrodes and the work piece. For this type of heating, the current causing the temperature increase corresponds to a current imposed by the electrodes (Figure 6 (b)). The generators used deliver alternating current. Phase I is the rise in temperature of the plate, Phase II is to maintain the temperature of the plate. Beyond 90 seconds, the steel plate is quenched, which is not considered in this study.
The experimental system put in place by ArcelorMittal is visible in Figure 6 (a), it involves a steel plate 1.5x400x800 mm^3 and two copper electrodes. The input current is located in the upper electrode, the lower electrode acts as a support.



(a) (b)

Fig 6: (a) experimental device of resistive heating on thin steel plates, (b) applied current input

The temperature is measured at four different points of the heated platen (Figure 7 (a)). Point 1 is at the interface between the electrode and the plate, points 2 and 3 are respectively 1 and 4 mm of the electrode and point 4 is at the center of the plate (Figure 7 (b)).



(a) (b)

Fig. 7: (a) temperature profile function of the distance to the electrodes (b) picture of the experimental device showing the position of the thermocouples TC

In this resistive study, it is assumed that there is no induction or diffusion effects. For such applications, it isn't needed to solve the full and costly Eddy Current problem but only necessary to calculate the conductors' resistance and Joule heating. Therefore the "resistive heating solver" has been used. Since no BEM is computed, very large time steps can be used which makes this solver very fast.
In this studies, the properties of the material (heat capacity Cp, thermal conductivity λ as well as electrical conductivity σ) are temperature dependent. The values of these parameters for the steel studied are shown on Figure 8.

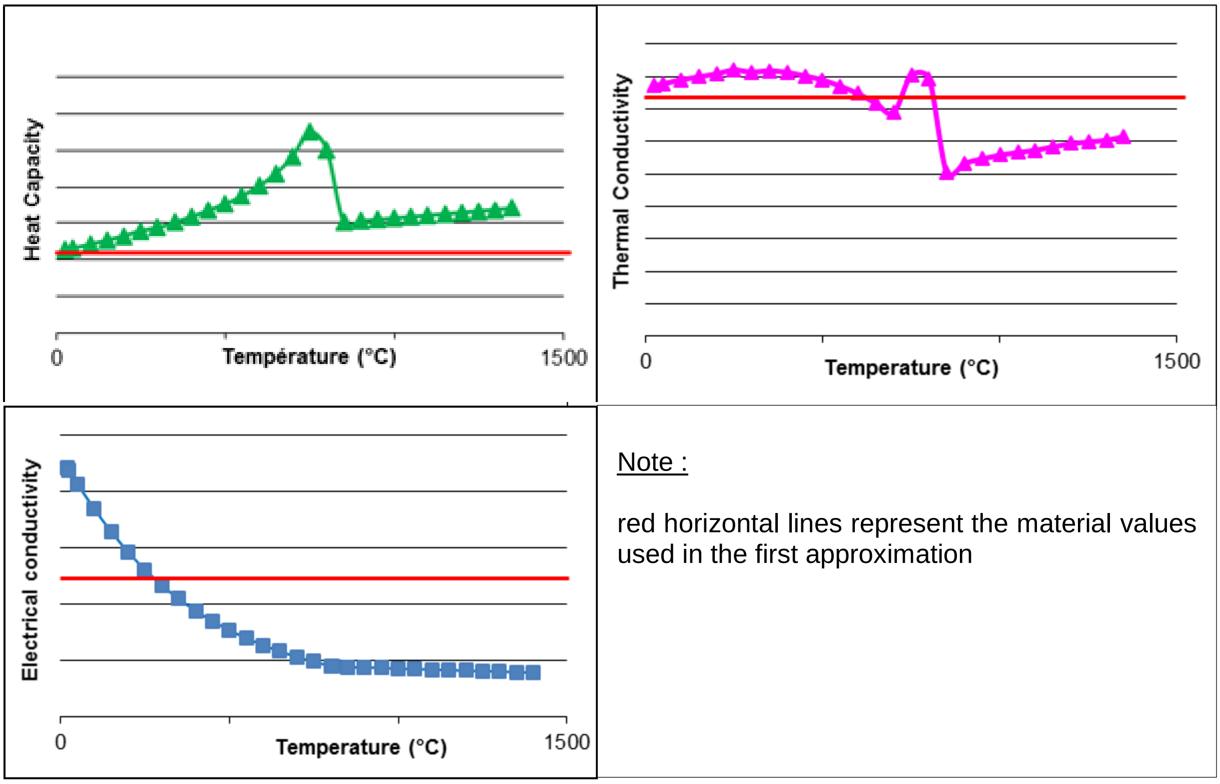


Fig. 8: properties of the material (heat capacity Cp, thermal conductivity λ as well as electrical conductivity σ) as function of temperature

3.2 Modelization of the industrial problem

3.2.1 Initial modeling assumptions

Figure 9 shows the choice of geometric modeling taken to represent the cases studied:

- the support electrodes are not modeled,
- In a first approximation, the contact between the electrodes and the plate is not modeled, thermal and electrical exchanges are therefore considered perfect. Nodes in contact between the electrodes and the plate are merged.
- The mesh size is 0.5 mm and there are five elements in the thickness.

The materials used are considered "rigid". The stresses and strains are not taken into account. Exchanges with the external environment, such as convection and radiation phenomena are initially neglected.

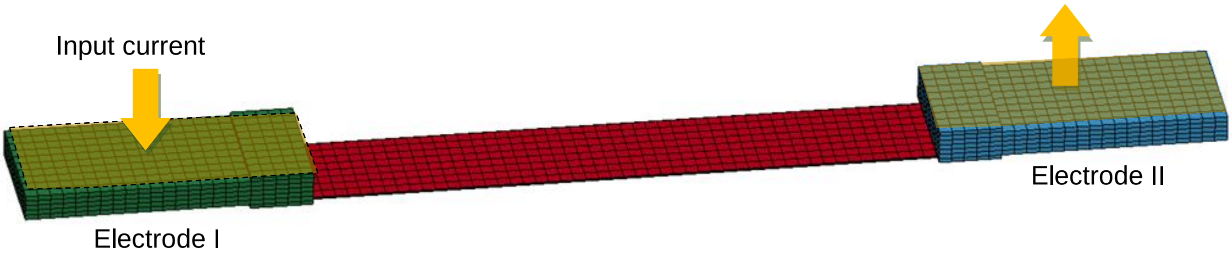


Fig. 9: modeling of the experimental device of resistive heating

Once the materials data implemented and geometry represented, the temperature profile at the thermocouple TC 4 (Figure 10 (a)) is visible in Figure 10 (b). With this particular implementation, the temperature exceeds 1400 °C to 80s. The plate get too hot during the temperature holding phase (phase II). To get closer to the experimental data in this phase, the phenomena of convection and radiation must be taken into account.

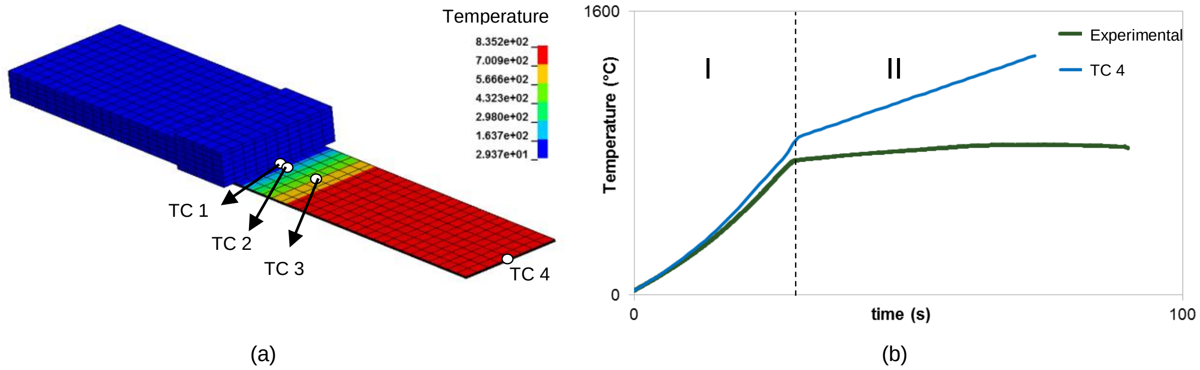


Fig. 10: (a) thermocouples position on the steel plate, (b) TC 4 temperature profile in the initial modeling conditions

3.2.2 Effect of convection and radiation

The convection phenomenon allows to take into account the heat exchange between the plate and the outside air in relatively low temperature ranges. The data used for convection coefficient are those present in the thermal Course LSTC [7] and [8] (Figure 11).

Beyond 700 degrees, the radiation phenomenon is predominant (on the order of 10 times superior to convection [7]). The values used for the radiation are linked to materials emissivity: 0.15 at 700 °C, 0.175 at 800 °C and 0.55 at 900 °C (Figure 11).

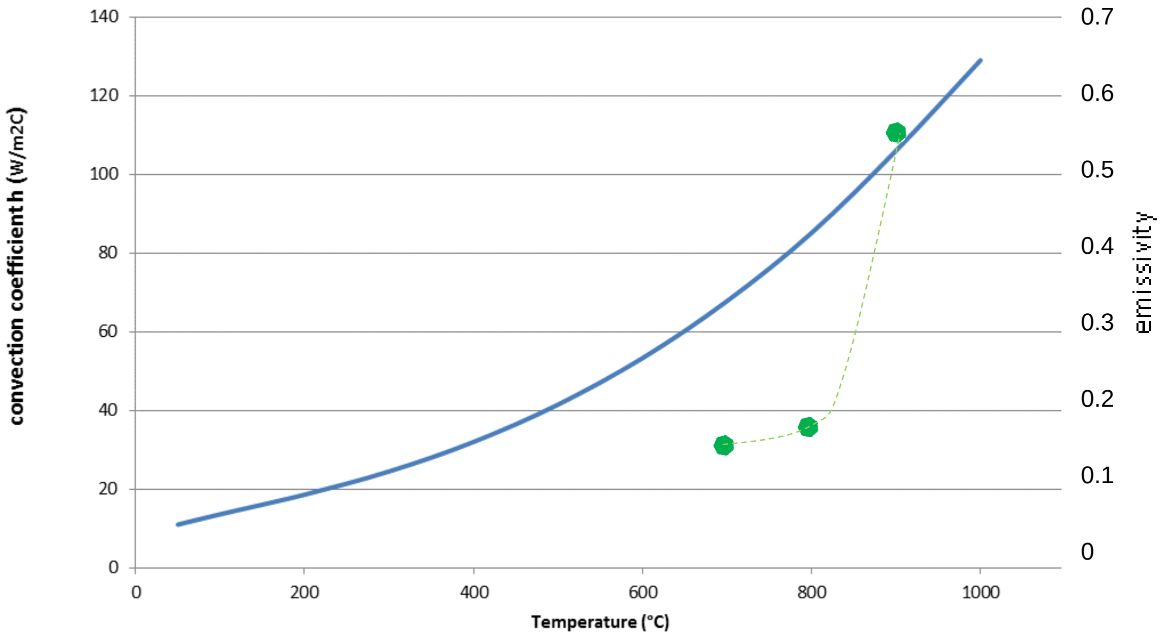


Fig. 11: evolution of the convection coefficient (blue) and emissivity (green) depending on the temperature

Figure 12 shows the temperature variations at the four points of measurement. Figure 12 (a) shows the evolution of the temperature profiles when the convection is taken into account. This implementation has correctly reproduce the experimental data. At the thermocouple TC 4, the temperature profile up to 60s is substantially superimposed on the experimental profile. Beyond 850°C and during phase II, the lightweight experimental temperature drop is not reproduced (area surrounded). To get closer experimental data, the radiation phenomenon was introduced.
Figure 12 (b) shows the evolution of temperature profiles when both the convection and radiation are taken into account. A very good correlation of the temperatures at the thermocouple TC4 is observed. However, other data points do not exceed the temperature of 700 ° C, they are not affected by the radiation.
During Phase I, the increase in temperature at the thermocouples TC1, TC2 and TC3 are not perfectly superposed on experimental measurements, but the monotony of the curves is observed. During Phase II, it appears notably strong difference in behavior at the TC3 thermocouple. There is a difference of 200 °C between experimental and simulation at 90s.

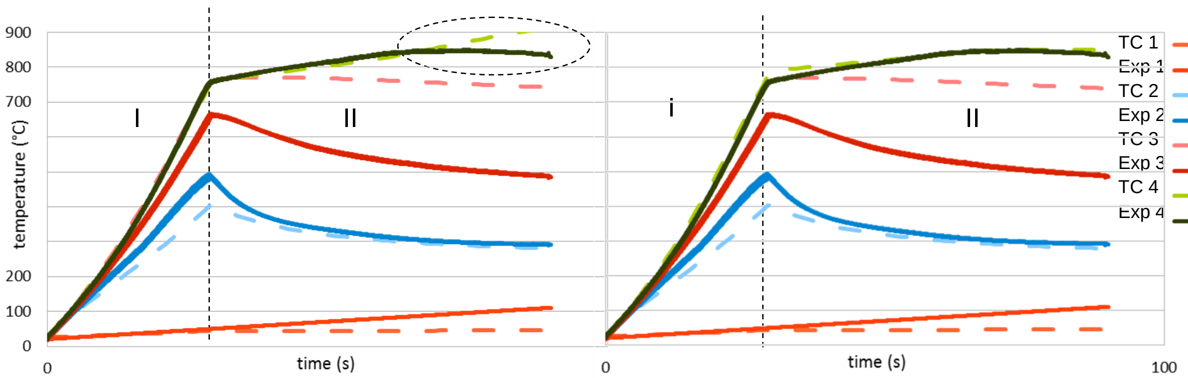
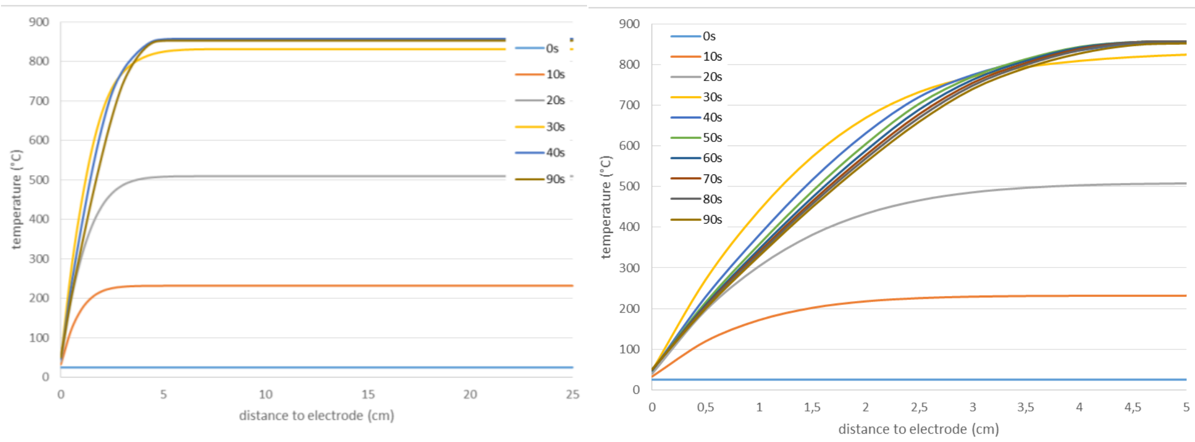


Fig. 12: (a) Temperature profile in function of the distance to the electrodes with consideration of the convection, (b) temperature profile as a function of the distance to the electrodes with consideration of the convection and radiation

We then sought to understand the phenomena involved in the vicinity of the electrode area Figure 13 (a) shows that beyond 5 cm, the temperature curves as a function of the distance to the electrode are fairly constant... Figure 13 (b) highlights the strong temperature gradients observed in the vicinity of the electrode area. Indeed, we see that a distance of 0.5 cm in this area can lead to temperature differences of more than 120°C. Figure 14 also illustrates this phenomenon: temperatures from modeling at a distance of 3 cm from the electrode are more similar to the experimental results at 4 cm corresponding to the thermocouple TC3.



(a) (b)

Fig. 13: (a) temperature variation as a function of the distance to the electrode, (b) temperature gradients in the vicinity of the electrode area

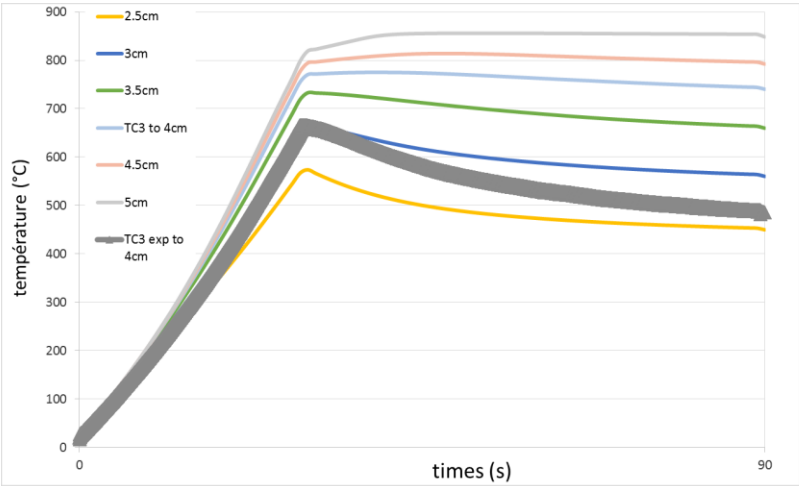


Fig. 14: numerical temperature variation as a function time for various distance from the electrode

These last results (Figure 13 (b) and Figure 14) illustrate the importance of taking into account the boundary conditions in the region of 0 to 5 cm of the electrodes...In this preliminary study, several assumptions were made:

- The data used for conductivity and emissivity coefficients are from the open literature,,
- The thermal and electrical contacts between the plate and the electrode are considered perfect,
- The conductive welding points to position thermocouples are not taken into account, their positioning has an accuracy of ± 2mm.

An improvement in the presented results certainly involve taking into account of these different phenomena.

4 Conclusion

Two alternative heating techniques were studied in this paper: the inductive heating which heats a steel plate by induced heating, without contact and the resistive heating that, thanks to two electrodes in contact, is used to apply a potential difference and heat the part by Joule effect. These studies show that the LS-DYNA software is able to simulate the inductive and resistive heating phenomena with an acceptable computation time in an industrial context and good accuracy.
Changes in temperature in a steel plate massively heated by induction seem physical. However, new experimental tests should be established for further correlation and validation of the model.
The resistive model is well correlated temperature profiles away from the boundary conditions. The current level of modeling is sufficient to represent the overall behavior of the part, but depending on the level of requirements requested by the manufacturer, the model can be improved. Several ways of improving were identified as the set-up of electrical and thermal contacts, mesh refinement and adjusting convection and radiation coefficients to the experimental conditions.

5 Acknowledgment

Dr. Romain CANIVENC of ArcelorMittal Global R&D Montataire is acknowledged for providing experimental results. Dr. Arthur SHAPIRO, Dr. Pierre L'EPLATTENIER, M. Iñaki ÇALDICHOURY and Mrs. Julie ANTON of LSTC are acknowledged for their helpful comments and suggestions.

6 Literature

[1] J. C. Nedelec, A new family of mixed finite element in R3, Num. Math, 50(1986), 57-81

[2] P. L'Eplattenier, I. Caldichoury, Electromagnetism module presentation, LS-DYNA-class notes, January 2014

[3] P. L'Eplattenier, C. Ashcraft, I. Ulacia, An MPP version of the electromagnetism module in LS-DYNA for 3D coupled mechanical-thermal-electromagnetic simulation, 11th international LS-DYNA User Conference, 2010

[4] L'Eplattenier P., Caldichoury I., "Electromagnetism and Linear Algebra in LS-DYNA", LSTC-LS-DYNA-EM-THE-1.1-1, Janvier 2012, 30 pages

[5] L'Eplattenier P., "Introduction of an electromagnetism Module in LS-DYNA for Coupled Mechanical-thermal-Electromagnetic Simulation", LSTC 10.2374/SRI08SP152, September 2008, 8 pages

[6] P. L'Eplattenier, I. Caldichoury, EM THEORY MANUAL: Electromagnetism and Linear Algebra in LS-DYNA, January 2012

[7] Shapiro A., "Using LS-DYNA for Heat Transfer & Coupled Thermal-Stress Problems", LSTC, 2013

[8] Shapiro A.: "LS-DYNA Features for Hot stamping", 9th International LS-DYNA Users Conference, 2006, 12-20 pages