

Simulation and Experimental Analysis of Heat Treatment for Steel Tie Rods: A Review

Kunal Kumar Singh^{1*}, Girish M. Dhote²,
Chetan Mahatme³, Prashant Kamble⁴

Abstract

The study is related to the actual heat treatment and simulation of the physical deformation of the steel tie rod and compares the observations for process optimization. The general methodology is to go for either traditional or induction hardening after getting the chemical composition of the material, followed by tempering, which is a key technique for improving the mechanical properties of materials, strength, hardness, etc., and to release stresses. This technology is widely used in the fields of manufacturing, aerospace, and robotics. This study is done in the current manufacturing scenario, which faces many rejections due to physical deformation leading to loss of money and time, so simulation aids in heat treatment are done before the actual process is done in a furnace for reducing rejection. Introducing computer-aided engineering (CAE) with the process of heat treatment is quite challenging and complex; hence, optimizing processes of heat treatment, a finite element analysis software for heat treatment, like COMSOL and Analysis System (ANSYS), is used for simulation. The boundary conditions and parameters in conventional gas carburizing heat treatment are studied and will be used in simulation software for analysis purposes. This will help the manufacturer to analyze the process to select appropriate heat treatment process parameters and hardness as per the application to avoid wastage. Moreover, integrating experimental data with simulation enhances the accuracy of predictive models and enables real-time adjustments. This approach not only improves product quality but also contributes to sustainable manufacturing by reducing energy consumption, minimizing rework, and extending component life.

Keywords: Heat treatment simulation, gas carburizing, heat treatment furnace, deformation, hardening

*Author for Correspondence

Kunal Kumar Singh
E-mail: kunal.cool1@gmail.com

¹Student, Department of Mechanical Engineering,
Yashwantrao Chavan College of Engineering, Nagpur
Maharashtra, India

^{2,4}Assistant Professor, Department of Mechanical Engineering,
Yashwantrao Chavan College of Engineering, Nagpur
Maharashtra, India

³Lecturer, Department of Mechanical Engineering,
Yashwantrao Chavan College of Engineering, Nagpur
Maharashtra, India

Received Date: May 20, 2025

Accepted Date: June 07, 2025

Published Date: June 19, 2025

Citation: Kunal Kumar Singh, Girish M. Dhote, Chetan Mahatme, Prashant Kamble. Simulation and Experimental Analysis of Heat Treatment for Steel Tie Rods: A Review. International Journal of Mechanical Dynamics and Systems Analysis. 2025; 3(1): 43–51p.

INTRODUCTION

This study aimed to analyze and optimize the design process so that heat treatment parameters can lead to deformation and topology optimization, while exploring the feasibility of various materials and operating conditions. The research simulated thermal and deformation behavior using a computer model and compared the results with experimental data by using heat treatment simulation analysis for thermal stress, deformation, and cracks, which can be identified before the actual process to introduce computer-aided engineering (CAE) with the process design of heat treatment to determine the correct process and hardness value required. To simulate the Multiphysics coupled processes of heat treatment, a finite element analysis software for heat treatment processes called COSMOL and ANSYS,

added on with Dante, is introduced, which integrates various numerical solving methods. This multi-solver approach provides robust computational support for addressing the heat transfer, deformation, and phase transformation issues encountered in various heat treatment processes. Similar to other finite element software, using COMSOL for heat treatment simulations requires inputs, including part model files, process parameters, and material property parameters. The preparation and organization of these data are complex with the use of heat treatment process simulation software.

INTELLIGENT SYSTEM FOR HEAT TREATMENT PROCESS AND PARAMETERS

Austempering heat treatment of ductile iron: Computational simulation and experimental validation aims to simulate the three-step Austempering heat treatment process of ductile iron parts using a coupled thermo-mechanical-metallurgical computational model. The objective was to predict the temperature evolution, dimensional changes, minimum required times for heat treatment stages, and phase transformations.

- *Model approach:* Thermal, mechanical, and metallurgical phenomena were simulated using finite element methods (FEM) at macro- and microscale levels.
- *Microscale modeling:* Metallurgical changes were modeled through representative volume elements, considering microstructure features.
- *Experimental validation:* The model's predictions were tested against experimental heat treatment data, analyzing the temperature evolution, phase transformations, and dimensional changes. The model successfully simulated complex heat treatment phenomena with good agreement with experimental results [1].

In a study published in the Journal of Applied Science and Advanced Technology on the topic of Hardness Analysis of Bearing on Heat Treatment Process in 2020, explained the objective of the study, which investigated the effect of the heat treatment process on the hardness of the inner and outer rings of bearings made from AISI 52100 steel. It aims to achieve the desired mechanical properties, specifically within the hardness range of 62-64 HRC, in accordance with the factory standards. Heat treatment processes, including hardening and tempering, were applied to the bearing rings to alter their mechanical properties. Following the treatment, the hardness of the inner and outer rings was assessed using the Rockwell hardness test. A Digital Rockwell Hardness Tester was employed with a load of 60 kg to ensure accurate hardness readings. Thus, we can conclude that the heat treatment process successfully achieved hardness levels within the desired range, ensuring that the bearings met the factory quality standards. For the outer ring, an average hardness of 63.5 HRC was observed, while the inner ring displayed an average hardness of 63.6 HRC. This process improves the material properties, making the bearings resilient and suitable for further machining and use [2].

Pedro Maranhão in the study on modelling and simulation of the induction hardening process: Evaluation of Gear Deformations and Parameter Optimization (2024). The objective of this study is to model and simulate the induction hardening process applied to gears, with the goal of understanding and optimizing its effects on gear deformations and overall performance, by developing a coupled electromagnetic-thermal-mechanical simulation model of the induction hardening process using suitable software tools (e.g., ANSYS, COMSOL, or other finite element analysis (FEA) platforms) to (a) analyze the thermal behavior during the heating and quenching stages to predict the temperature distribution within the gear. (b) Evaluation of the induced mechanical stresses and deformations resulting from thermal gradients and phase transformations. (c) After hardening, check for dimensional changes and gear distortion to ensure that they remain within the allowable bounds. (d) Investigate the influence of key process parameters such as coil geometry, power input, heating time, and quenching rate. (e) Parametric optimization is performed to minimize undesirable deformations while achieving the desired hardness profiles. (f) Validation of the simulation results with experimental or reference data, where possible, to ensure accuracy and reliability of the model [3].

Sun and Zhang employed the following methodology in their study: Development of an Intelligent Design and Simulation Aid System for Heat Treatment Processes Based on LLM (2024): Select a base Large Language Model (LLM), such as large language model meta AI (LLaMA), OpenAI GPT, or a comparable one. Fine-tuning or application of domain-specific engineering techniques to improve material-specific recommendations. Interpretation of the thermal profiles, phase transformations, and simulation outputs. The objective of this study is to develop an intelligent, AI-driven design and simulation aid system for heat treatment processes, leveraging the capabilities of LLMs to enhance decision-making, process optimization, and simulation support. The specific aim is to integrate LLMs (e.g., GPT-based models) into a design-assistance framework to support engineers in selecting appropriate heat treatment parameters based on material properties, part geometry, and performance requirements. The computer simulation of microstructure and properties (COSMAP) software was used for simulation and computational support to address heat transfer, deformation, and phase transformation issues encountered in various heat treatment processes [4].

The following goals investigation of crankshaft high-cycle bending fatigue based on electromagnetic induction quenching simulations and mean stress effects. (a) To precisely forecast the hardness profile and residual stress distribution in the crankshaft material, the electromagnetic induction quenching process was simulated. (b) Considering operational stress ranges and fatigue life prediction criteria, model the crankshaft's high-cycle bending fatigue behavior under realistic loading conditions. (c) The influence of mean stress on fatigue life was analyzed using methods such as Goodman, Gerber, or Smith-Watson-Topper (SWT) correction models. (d) The impact of induction hardening on fatigue performance was evaluated by comparing untreated and treated crankshaft models. (e) Simulation results are compared to experimental data or published benchmarks to ensure accuracy and reliability. The methods used were: (a) geometric modeling and material selection to develop a detailed 3D model of a representative crankshaft using computer-aided design (CAD) software and select the appropriate material commonly used for crankshafts (e.g., forged steel) and collect their mechanical, thermal, and electromagnetic properties. (b) Electromagnetic induction quenching in a multiphysics simulation tool (e.g., ANSYS, COMSOL) to model the induction hardening process [5].

The study of “the effects of oil quenching and over-tempering heat treatments on the dry sliding wear behavior of 25CrMo4 steel” (2024) is to measure and compare the hardness values of samples that have undergone oil quenching and over-tempering with untreated samples to better understand variations in hardness and enhance mechanical properties, and identifying methods to enhance wear resistance and durability of steel components used in industries like automotive and manufacturing. The material selection process, which focuses on 25CrMo4 steel, is commonly used in automotive components, such as axle shafts and turbine parts. The chemical composition of the steel was detailed, followed by sample preparation, in which cylindrical samples of the steel were cut into specific dimensions. Heat treatments (oil quenching and tempering) were used to obtain the desired microstructures and mechanical qualities. The heat-treated samples were preheated and then quenched in oil to induce martensitic transformation. The quenched samples were tempered at higher temperatures to decrease the martensite density and increase ductility [6].

An expanded FEM was implemented in COMSOL Multiphysics thermo-hydro-mechanical modeling of fluid flow in discontinuous porous media uses the COMSOL Multiphysics software tool, which is utilized for its flexibility in handling complex Multiphysics problems, including mechanical deformation, fluid flow, and heat transfer processes. Live Link for Matrix Laboratory (MATLAB): This integration allowed the researchers to use MATLAB functions for level-set analysis, extended finite element method (XFEM) physics enrichment, and pre-processing tasks, such as identifying enriched zones and updating crack geometries. Digital Rockwell Hardness Tester, which involves material hardness measurement, might have been used to determine the mechanical properties after treatments. Analytical Functions in COMSOL Built-in features like strain field and variable modifications were employed for XFEM-specific tasks. In this study, an extended finite element method (XFEM) framework for thermo-hydro-mechanical (THM) modeling of fluid flow in discontinuous porous media is developed and implemented in COMSOL Multiphysics [7].

The study on numerical simulation study on integral induction heating of backup roll aims to develop a FEM for electromagnetic-thermal coupling to simulate the integral induction heating process of backup rolls. The goal is to optimize heating parameters, improve temperature uniformity, and reduce costs associated with experimental trials. Methods are (1) A coupled electromagnetic-thermal FEM was developed using ANSYS. (2) Thermocouples and infrared thermometers were used to measure the temperature at different locations on the roller. The model provides a cost-effective tool for pre-evaluating induction heating processes and optimizing production parameters (Figure 1) [8].

The primary goal of the study on Material Modelling for Simulation of Heat Treatment was to develop reliable and efficient methods for simulating heat treatment processes using the FEM [8, 9]. Parameter optimization techniques for material model calibration are presented, using MATLAB-based tools. Stress-relief heat treatment processes are modeled by incorporating thermal boundary conditions derived from computational fluid dynamics (CFD) for convective heat transfer. Accurate boundary conditions, such as convective heat transfer coefficients, significantly affect simulation outcomes during the cooling stage (Figures 2 and 3) [9].

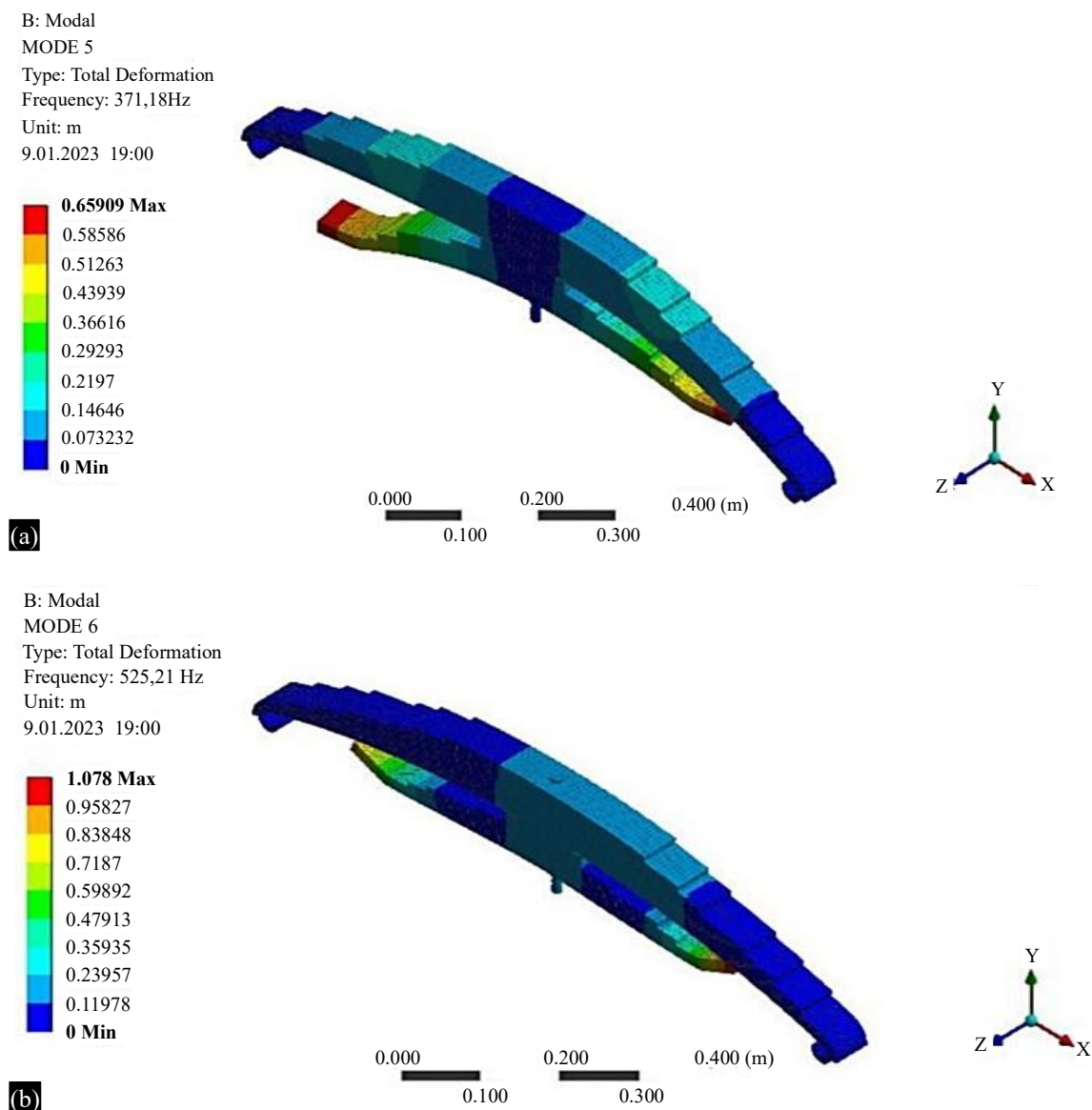


Figure 1. Simulations at different temperatures in ANSYS.

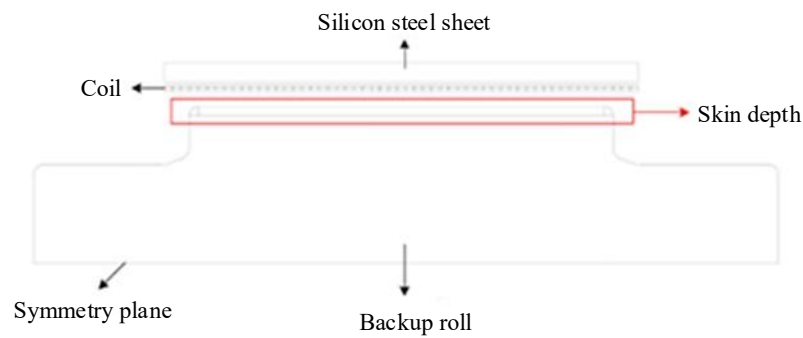


Figure 2. Schematic diagram of the FEM model.

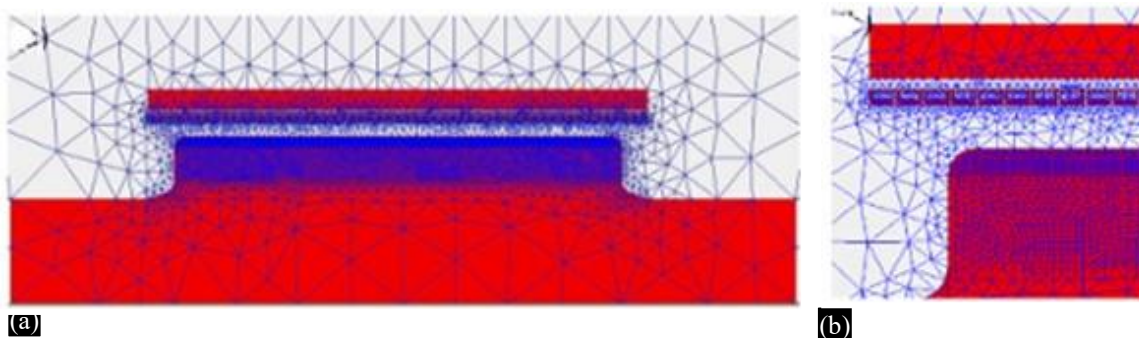


Figure 3. Mesh division in FEM model. The mesh division of the simulation modal (a) mesh of the whole modal; (b) localized mesh refinement at skin depth.

The goal of the literature review on heat treatment simulations of phase transformation, residual stresses, and distortion is to present a thorough analysis of numerical simulations of heat treatment procedures. The aim is to understand and predict phenomena such as phase transformation, residual stresses, and distortion, with a focus on improving manufacturing outcomes and reducing costly post-treatment machining owing to distortion. The simulation components described in the heat treatment simulation were divided into three major parts: temperature calculation, phase transformation, and mechanical response. Developing simulations based on physical models can reduce reliance on excessive experimental testing. Simulations are increasingly being used to predict and mitigate distortions in high-performance steel components subjected to heat treatment [10–14].

In the study on the numerical simulation of the induction heat treatment process of a shaft part, the purpose of using induction hardening and tempering was to examine the microstructure. The residual stress comparison results showed that the models and the experimental results fit each other quite well. Using the suggested numerical model, the effects of transformation and creep on the relaxation of tempering stress were further examined. The results of this study demonstrate that a coupled model that includes mechanical, thermal, transformation, and electromagnetic models for full induction heat treatment was examined. Using the suggested models, the induction hardening and subsequent tempering of the shaft part were simulated. Related experiments were conducted to verify the modeling results. In addition, a simulation was performed to study the tempering relaxation mechanism. The following conclusions were drawn from Figure 4. A uniformly hardened layer was obtained on the shaft after the proposed induction hardening process (Figure 5). Both volumetric shrinkage due to phase transformation and creep affect tempering stress relaxation [15, 16].

MODELING AND SIMULATIONS

In the study “a finite element simulation for carburizing heat treatment of automotive gear ring incorporating transformation plasticity” (2016), aim to develop a FEM to predict the deformation, temperature history, carbon diffusion, phase fraction, and hardness during the carburizing heat

treatment process for automotive gear rings. The model incorporates transformation plasticity to address issues, such as dimensional instability caused by phase transformations and thermal expansion/contraction.

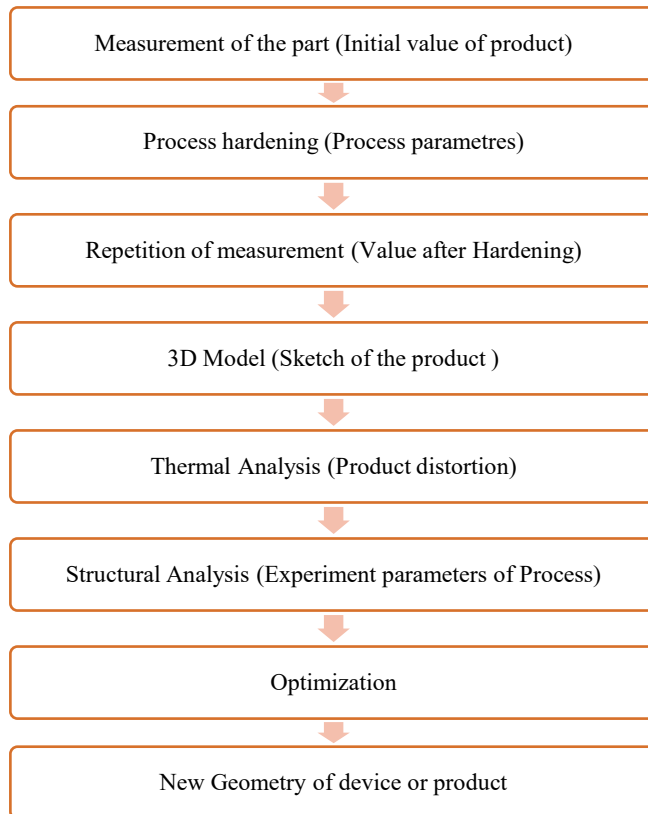


Figure 4. Flowchart of methodology.



Induction hardening equipment
Figure 5. Induction hardening machine.

Methods Used: Finite Element Modeling

Carburizing gas diffusion was simulated using Fick's law, considering temperature and pressure-driven diffusion. Phase transformations during the heating and cooling stages were modeled using Johnson-Mehl-Avrami-Kolmogorov (JMAK) and Koistinen-Marburger equations. The transformation plasticity was incorporated using constitutive equations for diffusional and diffusionless transformations. The thermomechanical responses, including thermal expansion and phase-related strains, were implemented in the model. The experimental validation was performed by conducting experiments on gear rings made of medium-carbon steel (SCR420). Microstructural and hardness analyses validated the predictions of the model. Thus, it can be concluded that model accuracy can be measured as the developed FEM successfully predicts the carbon concentration profiles, phase fractions, and hardness distributions after heat treatment. Transformation plasticity significantly influenced dimensional changes during carburizing and quenching. Higher carburizing temperatures result in deeper carbon diffusion and harder gear surfaces. Lower temperatures lead to an increased ferrite fraction and reduce dimensional instability [17, 18].

The objective of the study title "a study on the mechanical properties for developing a computer simulation model for heat treatment process" to develop a finite element simulation model to predict thermal deformation and strain-stress relations during heat treatment processes for High Strength Low Alloy (HSLA) steels. The methods used were as follows:

- *Material selection:* AISI 4120 and AISI 5120 steels were used for testing.
- *Heat treatment processes:* Martensite microstructure was achieved by heating to 850°C and quenching in oil, and bainite was processed via salt bath treatment at 500°C. Ferrite/pearlite was maintained at 650°C for stabilization.
- *Finite element analysis:* The data obtained from testing was used for simulations in ABAQUS to analyze heat transfer, thermal stress, and deformation during heat treatment, and it was concluded that the heat treatment successfully transformed the HSLA steels into martensite, bainite, and ferrite/pearlite phases, with detailed strain-stress relationships determined for each phase.

The study titled "*Modeling and Simulation of Quenching and Tempering Process in Steels*" aims to develop a modified model to predict microstructure evolution and hardness distribution during the quenching and tempering (Q&T) process for steels. This model focuses on achieving the desired mechanical properties, such as toughness and ductility, while minimizing cost and maximizing precision in simulations of phase transformations and tempering kinetics. *Methods:* Hardness distributions were predicted using experimental equations based on the iron-carbon phase compositions and alloy content. *Finite element method:* The COSMAP simulation code incorporates the transformation kinetics model. *Conclusions:* The model accurately predicted the phase distributions, hardness values, and microstructural evolution during the Q&T processes. The final microstructures differed between the outer and inner surfaces of the components owing to the varying cooling rates, and tempering successfully reduced the residual stress caused by quenching. The predicted hardness and stress distributions were in good agreement with the measured values, validating the effectiveness of the model.

Numerical simulation of the induction heat treatment process of a shaft part involving induction hardening and tempering by Tong et al. has the objective of developing a numerical simulation model to analyze the induction heat treatment process, including both induction hardening and subsequent tempering, of a shaft part made of SAE 4140H steel. *Methods:* Numerical modelling was proposed, in which a coupled electromagnetic-thermal-transformation-mechanical model was proposed. *Simulation Setup:* Induction heating was followed by water spray quenching and furnace tempering at 200, 400, and 600°C. Material parameters, including thermal conductivity, specific heat, and phase transformation kinetics, were experimentally determined. *Conclusion:* Residual stress decreased with increasing temperature tempering. At 600°C, the quenched residual stress was almost eliminated, reducing it by 92.2%.

The findings in a research study “evaluation of heat transfer coefficient during heat treatment by inverse analysis” focuses on developing an inverse heat transfer formulation using a two-dimensional FEM. The objective was to evaluate the heat transfer coefficients during the heat treatment of cylindrical carbon steel specimens cooled by air and water. The method used in this study focuses on developing an inverse heat transfer formulation using the two-dimensional FEM. The aim was to evaluate the heat transfer coefficients during the heat treatment of cylindrical carbon steel specimens cooled by air and water. Cylindrical carbon steel specimens were subjected to air cooling (fan cooling) and water quenching from an initial temperature of 500°C. Thermocouples were installed in the specimens to measure temperature variations at various locations during the cooling process, and we conclude that there is a need for higher-temperature experiments to evaluate the coefficients of industrial processes involving carbon steel heated to 800–900°C. This suggests the need to explore a three-dimensional inverse analysis for more complex workpiece geometries undergoing non-uniform cooling.

GAPS AND RECOMMENDATION

No automation in the heat treatment process regarding handling and other observations is shown. In the case of induction heat treatment, an electromagnetic-thermal coupling FEM was developed to study the induction heating of the backup roll by considering the actual input current and variation in the magneto-thermal property parameters of the roll with temperature. A field test was conducted, and the simulation accuracy of the established FEM was verified. Software analysis is not used in many cases, and no comparison has been made between actual and software data. This study attempts to fill the gap related to predicting structural deformation due to thermal induction and topology optimization in toothed gears by developing a computational procedure and validating processes in the automotive sector. For the simulation, the addition of the polymer was not considered, leaving the improvement of the model for future studies. Only microstructural examinations, hardness measurements, and wear tests were performed for the tempering conditions of 25CrMo4 steel. The parameters required for heat treatment or the boundary conditions for the simulations were determined using COMSOL software. COMSOL, which has a sophisticated Multiphysics modeling capability, is not mentioned. In this software, a combination of available built-in physics interfaces can be incorporated with user-defined physics interfaces to investigate formidable Multiphysics problems (COMSOL). Three different physics interfaces that consider fluid flow, heat transfer, and mechanical deformation are involved in the THM coupling study of fluid flow in deformable porous media.

CONCLUSION

The conclusion of the attached study highlights the effectiveness of simulation techniques in optimizing the heat treatment processes for steel tie rods. This study demonstrates how computational tools, such as COMSOL and ANSYS, combined with experimental methods, can help analyze and predict deformation, stress, and other physical behaviors during heat treatment processes. By simulating thermal and mechanical interactions, manufacturers can optimize parameters to reduce rejection, enhance product quality, and avoid waste. The integration of CAE with heat treatment design is presented as a promising, albeit complex, approach to achieving this goal. This study emphasizes the potential of simulations to accurately predict outcomes, enabling manufacturers to select appropriate process parameters and hardness values tailored to specific applications. However, challenges such as data preparation and the lack of automation in certain heat treatment processes suggest that future work could focus on improving these aspects.

REFERENCES

1. Bosnjak B, Radulovic B, Pop-Tonev K, Asanovic V. Influence of microalloying and heat treatment on the kinetics of bainitic reaction in austempered ductile iron. *J Mater Eng Perform*. 2001;10:203–11. doi:10.1361/105994901770345222.
2. Harish S, Bensely A, Mohan Lal D, Rajadurai A, Lenkey GB. Microstructural study of cryogenically treated En 31 bearing steel. *J Mater Process Technol*. 2009;209(7):3351–7. doi:10.1016/j.jmatprotec.2008.07.046.

3. Liu H, Zhao J, Tang J, Shao W, Sun B. Simulation and experimental verification of die quenching deformation of aviation carburized face gear. *Materials (Basel)*. 2023;16:690. doi:10.3390/ma16020690. PMID: 36676427.
4. Kučerová L, Burdová K, Marková V, Zálabský J, Ťavodová M. Using the design of experiments methodology to evaluate the heat treatment of additively manufactured maraging steel. *J Mater Res Technol*. 2023;26:8688–703. doi:10.1016/j.jmrt.2023.09.117.
5. Venkatesan M, Devaraj DH. Design and analysis of composite leaf spring in light vehicle. *Int J Mod Eng Res*. 2012;2(1):213–8.
6. Rajan KM, Deshpande PU, Narasimhan K. Effect of heat treatment of preform on the mechanical properties of flow formed AISI 4130 steel tubes—a theoretical and experimental assessment. *J Mater Process Technol*. 2002;125–126:503–11. doi:10.1016/S0924-0136(02)00305-9.
7. Jafari A, Vahab M, Broumand P, Khalili N. An eXtended Finite Element Method implementation in COMSOL Multiphysics: Thermo-hydro-mechanical modeling of fluid flow in discontinuous porous media. [Preprint]. arXiv. 2021 Dec 22. doi: <https://doi.org/10.48550/arXiv.2112.11918>.
8. Barka N, Chebak A, Brousseau J. Optimization of hardness profile of bearing seating heated by induction process using axisymmetric simulation. *Piers Online*. 2011;7(4):316–20.
9. Liu CC, Xu XJ, Liu Z, Fem A. A FEM modeling of quenching and tempering and its application in industrial engineering. *Finite Elem Anal Des*. 2003;39:1053–70. doi:10.1016/S0168-874X(02)00156-7.
10. Wen, T., Qin, X., Lv, Z., Yang, P., Chen, W., Xie, J., & Liu, F. (2024). Numerical simulation study on integral induction heating of backup roll. *Case Studies in Thermal Engineering*, 57, 104338.
11. Burnett JA, Padovan J. Residual stress field in heat-treated case-hardened cylinders. *J Therm Stress*. 1979;2:251–63. doi:10.1080/01495737908962404.
12. Chen JR, Tao YQ, Wang HG. A study on heat conduction with variable phase transformation composition during quench hardening. *J Mater Process Technol*. 1997;63:554–8. doi:10.1016/S0924-0136(96)02682-9.
13. Lee I. Plasma nitrocarburizing techniques with quenching and tempering for manufacture of vehicle parts. *Korean J Met Mater*. 2015 Jun;53(6):398–405. doi:10.3365/KJMM.2015.53.6.398.
14. Assefpour-Dezfuly M, Hugaas BA, Brownrigg A. Fire resistant high strength low alloy steels. *Mater Sci Technol*. 1990;6:1210–4. doi:10.1179/mst.1990.6.12.1210.
15. Babu SS. Classification and mechanisms of steel transformation. In: Totten GE, editor. *Steel Heat Treatment Metallurgy and Technologies*. Taylor & Francis Group; 2007. p. 107–8.
16. Tong D, Gu J, Yang F. Numerical simulation on induction heat treatment process of a shaft part: involving induction hardening and tempering. *J Mater Process Technol*. 2018;262:277–89. doi:10.1016/j.jmatprotec.2018.06.043.
17. Fukumoto M, Yoshizaki M, Imataka H, Okamura K, Yamamoto K. Three dimensional FEM analysis of helical gear subjected to the carburized quenching processes. *J Soc Mater Sci Jpn*. 2001;50(6):598–605. doi:10.2472/jsms.50.598.
18. Tong D, Gu J, Totten GE. Numerical simulation of induction hardening of a cylindrical part based on multi-physics coupling. *Model Simul Mater Sci Eng*. 2017;25:035009. doi:10.1088/1361-651X/aa5f7c.