SSRG International Journal of Mechanical Engineering
ISSN: 2348-8360/ https://doi.org/10.14445/23488360/IJME-V12I2P104

Volume 12 Issue 2, 39-47, February 2025

Original Article

Numerical and Experimental Study of Heat Treatment via Quenching Using CFD Analysis

Hemant P. Horane[1], Babaso N. Naik[1*], K. Durga Hemanth Kumar[2], Pranav Charkha[3], Ramdas G. Biradar[4], Rahul Saini[5]

[1]*Department of Mechanical Engineering, Walchand College of Engineering Sangli, Maharashtra, India.*
[2]*Department of Mechanical Engineering, Sagi Rama Krishnam Raju Engineering College(A), China Amiram, Bhimavaram, Andhra Pradesh, India.*
[3]*Department of Mechanical Engineering, D Y Patil University Ambi Pune, Talegaon Dabhade, Maharashtra, India.*
[4]*School of Engineering & Technology, PCET's Pimpri Chinchwad University, Pune, Maharashtra, India.*
[5]*Department of Research, Labtech Innovations, Pune, Maharashtra, India.*

[*]*Corresponding Author : babaso.naik@walchandsangli.ac.in*

Received: 06 December 2024 Revised: 08 January 2025 Accepted: 07 February 2025 Published: 25 February 2025

Abstract *- This study studies the quenching method within heat treatment processes numerically and experimentally. Agitation, which plays a critical role in ensuring the uniform distribution of heat to reduce the chances of deformation and cracking, is emphasized. Quenching experiments were conducted adhering to established standards, and a simulation of the quenching process was carried out using the commercial software Ansys Fluent. It was found that the numerical model outcomes were in good agreement with those from the experimental results across the three distinct quenching phases: vapor blanket phase, nucleate boiling phase, and convection phase. The validated model was then applied to simulate agitation at various fluid velocities. Fluid velocities of 1 m/s, 2 m/s, and 2.2 m/s were investigated to ascertain the impact of agitation. A significant drop in the probe's temperature was observed when comparing conditions with agitation to those without (0 m/s). However, no notable change in the cooling rate was seen across the range of fluid velocities tested. It was further observed that quenching quality does not depend heavily on the cooling rate; rather, it mainly depends on uniform cooling throughout the fluid domain facilitated by agitation. Further investigations utilizing the numerical model to understand the effects of different parameters and quenching oils on the quenching process, without the need for conducting physical experiments, are to be conducted.*

Keywords *- Computational Fluid Dynamics (CFD), Agitation, Quenching, Simulation.*

1. Introduction

Quenching is a critical stage in heat treatment, where a high-temperature component is rapidly cooled using a cooling medium or quenchant [1]. Various quenching mediums are available, including water, oils, brine solutions, polymer-based oils, etc. These are used depending on the cooling rate needed [2]; quenching is the case where the family of phenomena depends on the cooling rate that determines the final features of the component. This process can change the different properties of hardness, microstructure, strength, toughness, ductility, etc [3, 4]. The quenching procedure is usually divided into three stages of heat transfer to take place: vapor blanket stage, nucleate boiling stage, and convection stage [5].

There are some studies investigating the sufficient heat treatment parameters [2], and few of them focus on developing different approaches in the field of Computational Fluid Dynamics (CFD) modeling simulating very well mobile heat treatment furnaces [2]. A simulation was performed to assess both the thermal behavior of the furnace and the heat treatment of the metal components. This type of furnace is typically used for heat treatment of impellers to get the necessary mechanical properties through stress relief. In practice, monitoring the temperature changes inside the metal pieces is very challenging; CFD simulation, a well-known tool for its accuracy, allows us to predict the temperature changes during the heat treatment process [6]. The CFD model is a versatile approach incorporating turbulent combustion, thermal radiation, and conjugated heat transfer with accuracy and reliability. [2]. Jan and MacKenzie (2024) developed a Computational Fluid Dynamics (CFD) simulation methodology for quenching processes aimed at controlling thermal residual stress, emphasizing the calibration of conjugate heat transfer models and optimizing air and water quenching methods for aluminum engine components [7]. They further extended the study regarding process parameters like pressure change in quenchant and equipment position. They also optimized the process to have the minimum losses in quenchant. Amine Kacem et al. have described the experimentation done for quenching of Inconel 718 with and without agitation by an impeller-based turbine in experimental setup and simulation for the same [8]. This work mainly focuses on the access effect of the agitation on quenching work. It is concluded that the agitation increases in cooling rate significantly in the boiling stage and convection stage, where the heat transfer is enhanced by the movement of fluid around the probe, but at the same time, this work has concluded that there is a very slight change in vapor blanket phase due to agitation in this work velocity measurement is done by particle image velocimetry. The agitation is provided from 410 rpm to 1000 rpm. Lastly, the article has suggested that agitation improves the quenching by creating uniform temperature distribution during the process [9]. D. Scott Mackenzie et al. demonstrated that contamination and oxidation of quench oil can significantly alter the maximum cooling rate and the temperature at which maximum cooling occurs [10]. This can lead to increased part distortion, cracking, and non-uniformity in material properties. To ensure the production of high-quality parts, a control program is essential for monitoring and tracking the performance of quench oil. Additionally, the author provides a detailed overview of the parameters used to quench oil quality.

Recent developments in quenching techniques and the use of Computational Fluid Dynamics (CFD) also shed light on the heat transfer details and provide information on design and process optimization. The development of CFD techniques for controlling thermal residual stresses in aluminum engine components was emphasized in several studies that attempted to optimize parameters related to quenching, such as pressure or placement of equipment [7-9]. Likewise, the effects of agitation on quenching by way of experimental rigs and CFD simulations show that agitation greatly boosts cooling rates throughout the boiling and convection stages but has little impact on the vapor blanket phase [8]. The work is based on water quenching using cylindrical impinging jets, and its improvements in cooling uniformity are presented. Also, exploring the effect of the contamination of quenching oils by oxidation on cooling rates further aids in ensuring the quality of oil for effective quenching. These studies complement foundational work, which gave a contemporary background to quenching practices and used CFD to predict performance in heat treatment furnaces [9-11].

Quenching is a key step in heat treatment, and rapid cooling significantly impacts mechanical properties like hardness, strength, and ductility of materials. Although much research has been done on heat transfer dynamics during quenching, most experiments have been performed under static conditions, ignoring the influence of agitation as a means to improve the homogeneity of the cooling process and minimize defects such as distortion and cracking. Furthermore, prior studies have usually focused individually on experimental or numerical techniques, with the integration of both not yet being done extensively to address the quenching behavior comprehensively. (2D) skyrmions are the main focus of this study, which aims to fill some of these gaps by using an experimental and computational approach to investigate how agitation affects quenching performance. With this objective, the study investigates the influence of changing liquid velocities on cooling rates and homogeneity, considering distinct stages of quenching, leading to optimize operational variables and obtaining guidelines for use in industries. Additionally, this systematic methodology provides valuable insights into the thermophysical mechanisms underlying heat transfer, paving the way for both more effective and improved quenching process efficiencies.

This study seeks to address the role of agitation in the quenching process using (i) verification of a Computational Fluid Dynamics (CFD) model with experimental data, (ii) investigation of the effect of varying fluid velocities on cooling rates and uniformity during the quenching phases, and (iii) the optimization of operational parameters to minimize defects like distortion and cracking. Furthermore, the research aims to yield practical knowledge for industrial use, paving the way for better and more dependent quenching processes. The outlined objectives also fill in the gaps of current research and have a specific goal where the present study combines experimental validation with advanced simulations to improve the significance of the research.

1.1. Scope for Present Research Work

Agitation during quenching is the process of providing motion to fluid to ensure efficient heat transfer. There are multiple reasons for distortion and crack formation during quenching; however, one of the major reasons is the prolonged vapor blanket stage and poor heat transfer in the convection stage.

Agitation ensures uniform distribution of quenchant oil in quenching tanks [11]. The present work simulates the quenching process computationally using a commercial solver, Ansys Fluent and validates the results with experiments. Further, it investigates the effect of fluid velocity on the quenching process.

2. Materials and Methods

This study utilized Industrial quenching oils - Determination of cooling characteristics - Nickel-alloy (Inconel 600) as the material for the testing probe, chosen for its high-temperature resistance and relevance in industrial applications. The probe dimensions were 400 mm in length and 240 g in weight, with a K-type thermocouple integrated to measure temperature during quenching.

The quenching medium used was a specialized quenching oil, preheated to 80°C, adhering to the manufacturer's specifications. Standard Test Method for Determination of Cooling Characteristics of Quenching Oils by Cooling Curve Analysis, specifically ASTM D 6200-01 and ISO 9950:1995, were followed to ensure uniformity and accuracy in the experimental setup. As per these standards, the probe must be prepared, heated, and immersed in the oil under defined conditions, such as pretreatment of the probe with emery paper and oil purity to evade any contamination.

The quenching vessel was 1.8 L, and the vessel design ensured uniform distribution [12]. Finally, cooling curves at 1 m/s, 2 m/s, and 2.5 m/s have been simulated and plotted in order to study their behavior [13]. Such settings were essential to gain reliable and repeatable results and also represented a solid base for validating the numerical simulations.

2.1. Properties of Quenching Fluid

Apart from the agitation, some parameters are important and affect quenching quality. Studying the factors influencing the quenching process is essential to improve efficiency and prevent undesirable outcomes.

The important parameters are viscosity, flash point, water content, contamination and sludge formation. Sludge formation, which poses significant hazards, is primarily caused by the oxidation of quench oil and localized overheating, or "frying," of the oil.

The relative amount of sludge in quench oil can be quantified and expressed as a precipitation number, determined according to ASTM D 91. An estimate of the oil's remaining service life can be made by comparing the tendency for sludge formation between fresh and used oil. The parameters mentioned above should be regularly tested in oil samples according to ASTM standards [14].

2.2. Experimental Methodology

The test setup used for experimentation is IVF Smart Quench II [8]. With the help of this setup, cooling curves are plotted for various cases. The test setup consists of the furnace with dimensions 215 x 295 x 305 mm. The weight of the furnace was 6.9 kg, and the approximate power consumed by the furnace was 650 watts. A quenching vessel of 2 litres capacity and 115 mm in diameter, 200 mm in height and 5 mm in thickness. Its position was controlled by a stop ring for vertical alignment. A testing and supporting probe of weight 240 grams and length 400 ± 5 mm was used. The material for testing and supporting the probe was Inconel 600. A thermocouple of K type with a bid diameter of 1.5 mm was used to record the temperatures. The thermocouple output at specified time intervals was stored using a Data Acquisition System (DAS) QDA1032B-THC (Make: Quazar Technologies).

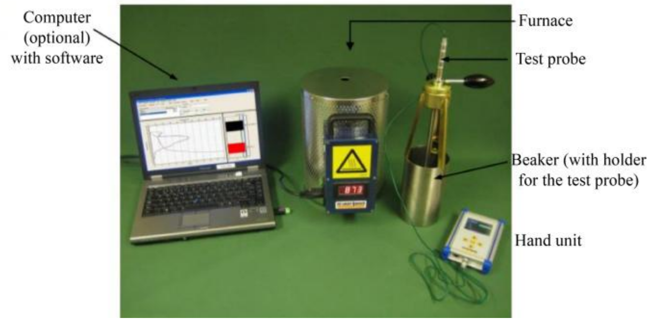


Fig. 1 Experimental set-up

The experiments were conducted by preparing the probe, ensuring the removal of oxidation from previous tests and cleaning it thoroughly with emery paper to prevent impurities that may affect the cooling curve. Direct contact was avoided with the cleaned probe to prevent fingerprints, which could introduce surface irregularities and influence the test results. The furnace is then started, and the probe is heated to 855 °C, although the cooling curve is plotted from 850°C to account for potential heat loss to the atmosphere; hence, there was a 5°C allowance. Simultaneously, the quenching vessel was filled with 1.8 litres of testing oil, ensuring no impurities were present to avoid contamination, and the oil was preheated to 80 °C per the manufacturer's specifications. Once the probe reaches the target temperature, it is removed from the furnace and immersed in the quenching vessel, allowing it to cool for 60 seconds. Upon completion, the results are saved using the data acquisition system, with two plots generated: one for the temperature drop per second and the other for the cooling rate, representing the ratio of a temperature difference between successive time intervals to the time interval itself.

2.3. Numerical Simulation

The need for simulation arises due to heat treatment experiments' time-consuming and costly nature. Each parameter variation, such as fluid velocity, temperature, and cooling rate, requires separate experimentation, whereas, in CFD, these variations can be easily adjusted and analysed in a shorter time. The CFD better reduce the experimental validation, thus better improvement [13]. CFD also allows the simulation of fluid mechanics around quenching components and, thus, the optimization of processes. Evaluating fluid behaviour, with the help of CFD simulation, around the testing probe during the quenching process is very important. Offers the possibility of scenario building and parameter tuning. The simulation is done by solving differential equations based on the conservation of mass (also known as continuity equation), momentum and energy by discretizing the governing equations. Turbulence modeling is employed in this simulation, provided by the ANSYS FLUENT software package, which uses the element-based finite volume method to guarantee the conservation of momentum, mass, and energy. In this approach, control volumes are considered for every node of the equation.

The first step would be the pre-processing stage, followed by the solver step, and finally, the post-processing stage, which details the key steps in CFD analysis and explains how each one can be applied to the CFD modelling of the current simulation. Thermal simulations using ANSYS FLUENT. The heat is exchanged from hot solid components to the surrounding quenchants during quenching [8]. The difference in temperature between components causes both the positive (hardening, etc.) and negative (hot cracking, etc.) phenomena of change in the material's properties, deformation and residual stresses. This section simulates heat transfer in different quenching processes and determines the cooling curve. These include CAD modelling, meshing, solver and post-processing.

2.4. Validation of the Simulation and Reliability Analysis

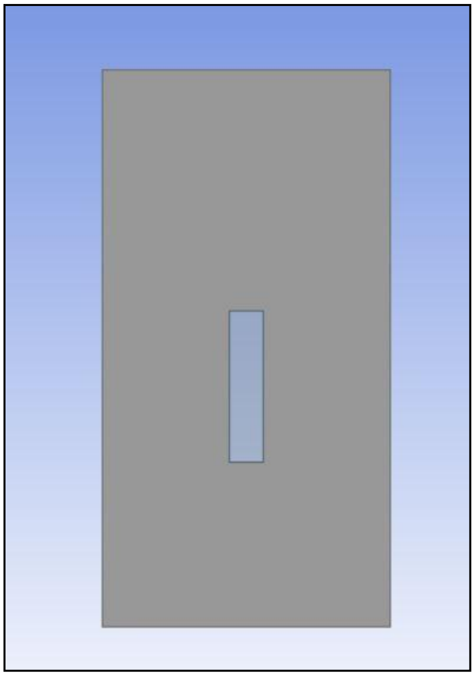
Detailed error analysis and statistical correlation coefficients further confirm that the numerical simulations match the experimental results. The average percentage error of the temperature values between the experimental and numerical models was calculated to be 3.95%, indicating the numerical model is very accurate. Moreover, correlation coefficients have been calculated statistically to represent the data of the strong common trend, and it was reported that the high positive correlation of all focused datasets confirms that the results from the CFD analysis are reliable.

The fault analysis lays out the model's accuracy in predicting temperature profiles during different stages of the quenching process with small errors. These results confirm that the CFD model accurately captures complex heat transfer phenomena, especially with agitation effects. This thorough validation increases the reliance on the model for optimizing operational parameters and simulating conditions infeasible through physical experimentation.

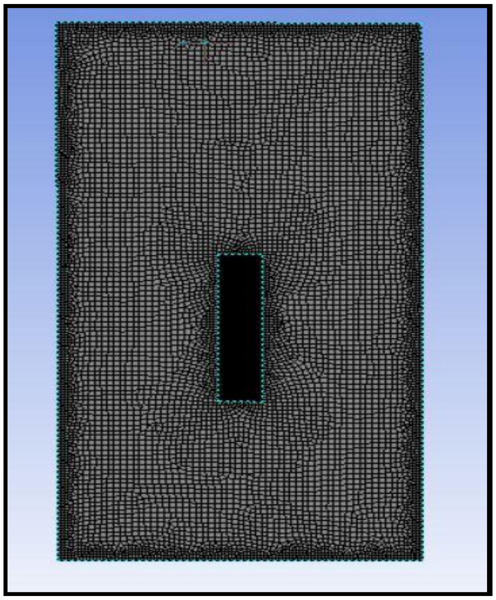
2.5. CAD Modelling and Meshing

CAD modelling is the first stage of a CFD simulation, simplifying the system and omitting unnecessary parts. Removing unnecessary parts minimizes the complexity of the solution and saves computational time [15]. This makes it clearer and more interpretable. The CAD model was simplified for this study by eliminating components that did not affect results, including the support structure, stop ring, and probe holders.

Next, this CAD model was meshed into a mathematical representation. This griding or division of the domain or system is called meshing to solve the partial differential equations that need to be solved [16]. Critical areas of the system, such as the fluid domain around the testing probe, were prioritized during meshing. A dense (fine) mesh was applied around the test probe, while a coarse mesh was used for less critical areas [17]. This combination of fine and coarse meshes effectively reduces computational time while maintaining accuracy in the simulation.



(a) Simplified CAD model of experimental setup



(b) Meshing of CAD model
Fig. 2 CAD model

At this stage, the boundary conditions and material properties extracted from experimentation are inserted to set up the CFD model as per requirement. This is then solved in the solver to obtain the desired results.

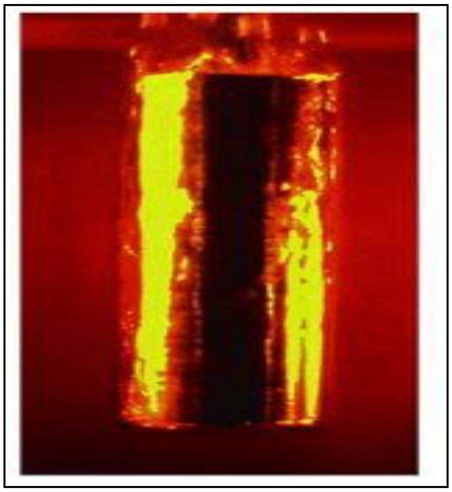
In this case, there is an interest in the behaviour of the test probe, specifically its temperature profile. Therefore, the solution in the solver has been set up to acquire the temperature profile of the test probe, considering variations in agitation. Table 1 presents the necessary settings applied in the solver.

Table 1. Settings applied in the solver

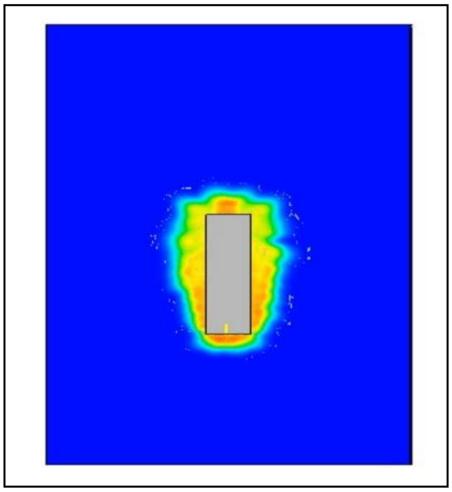
Model	Settings
Space	Two-dimensional simulation
Viscous	Standard k-ε turbulence model with scalable wall functions
Wall Treatment	Enhanced wall treatment (suitable for low y+ values)
Heat Transfer	Conjugate heat transfer (CHT) enabled for solid-fluid interactions
Multiphase	Volume of Fluid (VOF) method for phase interactions

3. Results and Discussions

The results of the experimental work and the numerical simulations are presented in this section. Standard ASTM methods were used for the quenching experiments, and commercial software was used for the numerical simulations. A detailed comparative study between the experimental and simulation results is given. The results of this analysis could provide important guidance for optimizing process parameters, especially with respect to agitation [18]. Quenching is subdivided into three different cooling stages: vapor blanket, nucleate boiling, and convection, with each range contributing different mechanisms to cooling and thus impacting material properties substantially [19]. During the vapor blanket phase, a vapor layer insulates the quenchant from the heated component, delaying contact and heat transfer into the quenchant. A detailed temperature profile during this stage highlights how significant bringing in agitation to eliminate the vapor layer is in speeding the transition to the nucleate boiling phase. In the nucleate boiling regime, local sites develop vapour bubbles that enable the formation of regions of increased heat transfer. In this stage, a steep drop in surface temperature is observed, as agitation helps displace the bubbles and enhances cooling efficiency. The last phase, the convection phase, is here a gradual and uniform cooling rate. During this stage, the temperature distributions serve as a reference as the uniformity of distributed fluid is achieved, which, in turn, serves to minimize the temperature profile, reducing distortion and cracking of the workpiece due to the stagnation of the internal fluid. The implications of these temperature profiles on material properties, such as hardness, residual stresses, and microstructural uniformity, are significant and demonstrate the critical interplay between the quenching phases and agitation.



(a) Experimental vapour blanket phase



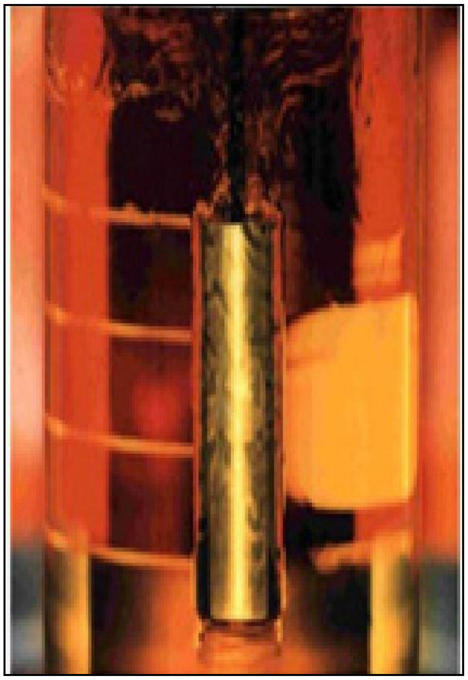
(b) Simulated vapour blanket phase
Fig. 3 Vapor blanket phase

4. Vapour Blanket Phase

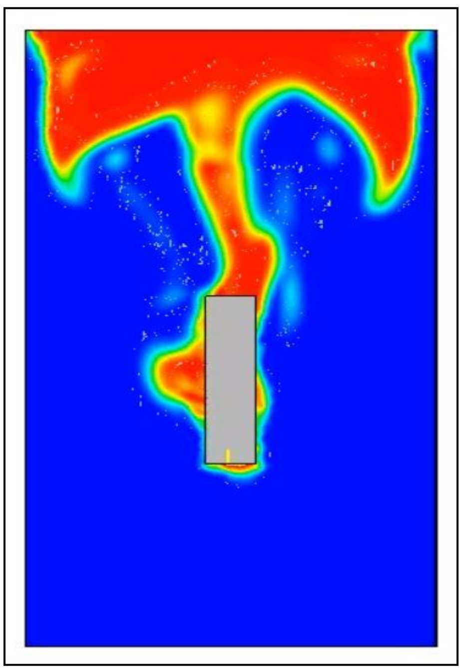
Figure 3 shows the experimental and simulated results for the vapor blanket stage. Figure number is the experimental photograph for the test, while figure number is the simulation result for the vapor blanket stage from Ansys Fluent. This is the first stage of quenching. When the component encounters the quenching oil, the temperature difference causes a vapor layer around it. The vapor acts as an insulator, preventing direct contact between the component and the quenchant. The same phenomenon can be seen in both the experimental results and the simulated results.

5. Boiling Stage

In Figure 4 the experimental and simulated results for the boiling stage are depicted. Figure 1 shows the experimental photograph for the test, while Figure 4 shows the simulation result for the boiling stage obtained from Ansys Fluent. This stage referred to as the transport cooling phase, is marked by the breakdown of the vapor blanket into bubbles or layers, which are subsequently displaced away from the component. This movement increases the contact between the quenchant and the component, leading to the achievement of the maximum cooling rate compared to other stages.



(a) Experimental boiling phase

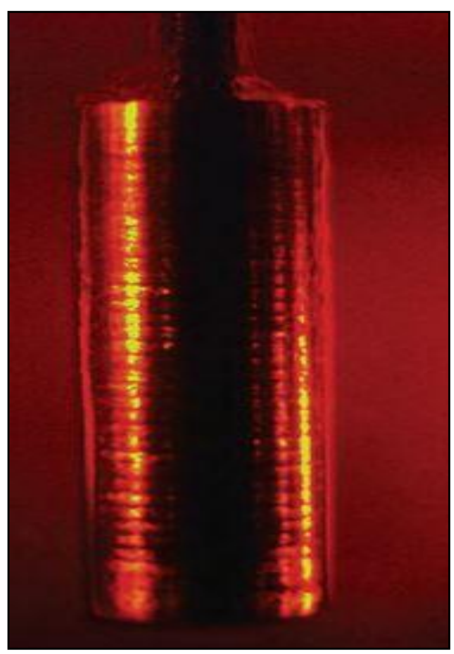


(b) Simulated boiling phase
Fig. 4 Boiling phase

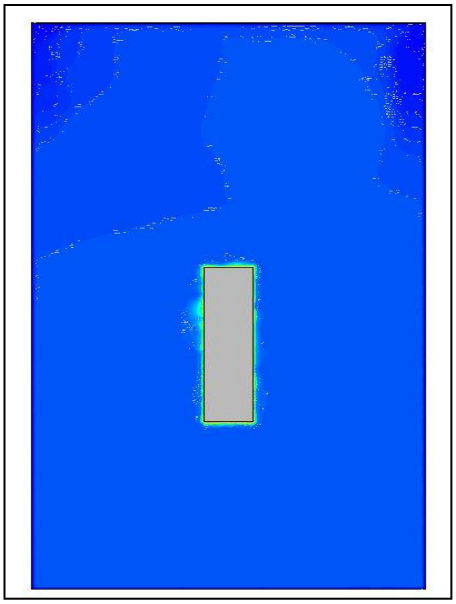
6. Convection Stage

Figure 5 shows the experimental and simulated results for the convection stage. Figure 5 is the experimental photograph for the test, while Figure 5 is the simulated result for convection from Ansys Fluent. This is the third stage of quenching, also known as the liquid cooling stage. This phase begins when the temperature of the component surface equals the boiling point of the quenchant. The cooling rate is slow compared to the other stages. Agitation (mixing) arrangement in the quenching tank Figure 5 presents both the experimental and simulated results for the convection stage of quenching, highlighting key observations and data obtained. Figure 5 (a) shows the picture obtained in the test, and Figure 5(b) presents the result of convective simulation in Ansys Fluent. The cooling stage is known as the liquid cooling stage and is a very important step that is recognized to be the third stage in the quenching process. Since the cooling process with water usually has a high heat transfer rate and is slow, the involucre with the quenching tank is used for the agitation (or mixing) arrangement to keep the energy out of the quench. Moving the quenchant is a strategic step to achieve a more uniform cooling action across the part surface and to significantly increase cooling speed. We need this to have the right properties to avoid thermal stresses that might lead to cracking or other material deformation. Computational Fluid Dynamics (CFD) simulations, such as those conducted in Ansys Fluent, are also essential to ensure that the outcomes of the quenching process are accurately predicted. Engineers can optimize parameters of the quenching process, such as the design of the agitation system, to compensate and achieve the desired cooling rate, homogeneity in the flow field and to minimize the formation of the critical cooling rate for each material through modelling fluid flow and heat transfer [7].

The quenching experiment is displayed for one of the minutes in Table 2. They measured the probe's temperature at multiple time points (i.e., 6, 12, 60 seconds). CFD simulation was used to skip temperatures at these same intervals. The two magnitudes are in excellent agreement. This validates the experimental results with a computational model of the quenching process.



(a) Experimental convection phase



(b) Simulated convection phase
Fig. 5 Convection phase

Table 2. The quenching process

Time Interval in Sec	Experimental Temperature Readings in °C	Simulated Temperature Readings in °C
0	850	850
6	733.271	726.97
12	427.12	420.82
60	184.42	178.12

7. Effect of Agitation on Quenching

Agitation, as mentioned in the previous sections, contributes greatly to increasing both the efficiency and quality of the quenching process. Agitation, the process of stirring or moving fluid around, facilitates efficient heat transmission and, thus, is vital in enhancing quenching treatments. The absence of agitations can result in prolonged stages of vapor blanket and poor convection heat transfer, leading to distortion, crack, or stains on the component, thus devaluing the product. Agitation is used for a few reasons to address these issues.

This promotes fluid motion first, interrupting the vapor blanket phase sooner. Second, it improves forced convection heat transfer in the convection stage of the quenching. This constant flow keeps some regions of the quench oil from staying in touch with the same piece, reducing strain effects. Agitation also prevents overheating of the oil in the hot-spot area, which would otherwise shorten the life of the quenchant. Additionally, agitation helps to achieve uniform temperature distribution of the quenchant oil around the quenching tank, ultimately resulting in a consistent quenching process [20].

In the present configuration, fluid agitation has been imparted at three controllable fluid velocities as 1, 2, and 2.5 m/s and specific temporal temperatures were numerically modelled; the cooling curves at each fluid velocity have been plotted. Agitation helps lower temperature by improving heat transfer via fluid turbulence. The action of agitation is more effective in disrupting the vapour blanket layer, resulting in direct contact with the heated surface from the quenchant. This mechanism accelerates heat dissipation during the vapor blanket and convection phases. Increased agitation also enhances forced convection, improving uniformity in temperature distribution throughout the fluid domain.

To provide greater clarity, a comparison chart summarizing the cooling curves at different fluid velocities (0 m/s, 1 m/s, and 2.5 m/s) has been introduced. The chart illustrates how increasing agitation significantly reduces the time for temperature drops, particularly during the convective stage, while maintaining a consistent cooling rate across varying velocities. This visual aid helps highlight the correlation between fluid velocity and quenching efficiency, offering a deeper understanding of the underlying heat transfer mechanisms.

Table 3. Significant temperature drop in temperature of the probe without agitation (0 m/s) and with agitation

Time Interval (Sec)	Temperature °C at Different fluid Velocity (m/sec) for Time Interval		
	0 (m/sec)	1(m/sec)	2.5 (m/sec)
0	850	850	850
6	733.27	721.67	705.77
12	426.12	415.52	399.62
60	183.42	172.82	156.92

It is observed from Table 3 that there is a significant temperature drop in the temperature of the probe without agitation (0 m/s) and with agitation. However, no noticeable change in the cooling rate across the range of fluid velocities was considered. It is further noticed that the quality of quenching is not strongly dependent on the cooling rate. It mainly depends upon uniform cooling across the fluid domain, which is achieved through agitation. The figure below presents the pictorial drop in temperature at various fluid velocities.

Also, based on the results presented in Figure 6, It is evident that the simulated results align with the trends observed in the experimental data with a few variations. Similarly, simulating the same model with agitation has resulted in the expected enhancement in convective stage temperature drop. While there is a minor change in the duration of the vapor blanket stage, heat transfer is notably improved due to agitation. The CFD model has produced the anticipated results outlined in the objective section.

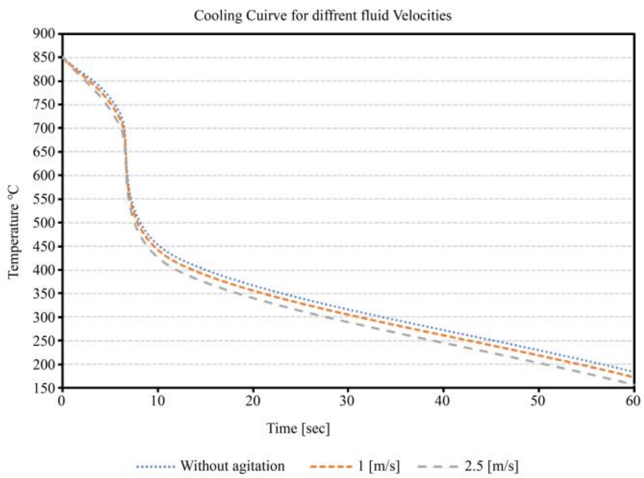


Fig. 6 Cooling curve for various fluid velocities

The superior results achieved in this study, particularly in terms of uniform cooling and reduced distortion during the quenching process, can be attributed to the combined experimental and numerical approach adopted. Unlike prior works that predominantly focused on static quenching processes or limited velocity ranges, this study introduces a detailed analysis of agitation-induced fluid dynamics at varying velocities. CFD simulations allowed for accurate temperature distribution and fluid movement around the probe, minimizing the need for physical experiments, which can be time-consuming. This means that including three different fluid velocities (0 m/s, 1 m/s, and 2.5 m/s) enabled additional information to be learned regarding the role of agitation in the different phases of quenching progressed. Results confirm that agitation improves the heat transfer rate by reducing the time of the vapor blanket and a convection heat transfer, which results in better cooling than previous techniques described in the literature. Additionally, the validation of Computational Fluid Dynamics (CFD) models with experimental data showed a substantial level of agreement (3.95% deviation), validating the robustness of the methodology.

8. Practical Applications

The results of this study have important implications for industrial heat treatment practice, particularly for enhanced quenching of high-performance components. The proven effects of agitation on uniform cooling can be translated into automotive, aerospace, manufacturing, and other industries where distortion and cracking, e.g. defects, can be minimized. It also results in gradually building knowledge within the developed models, which can then be used in an iterative fashion to optimize various operational parameters in each subsequent experimental quench. Moreover, the results may be leveraged to design experimental quenching systems, such as ones required for certain materials or complex geometries. The real application of this research has the potential to improve the operation of industrial heat treatment of parts both with respect to quality and time.

9. Conclusion

Agitation is very important in accelerating the quenching process due to improved heat dissipation and promoting adequate cooling. Experimental tests were carried out following ASTM D 6200-01 standards and correlated using numerical simulations; with a great concordance of the qualified data, an average variation of 3.95% was achieved. Changes made to the model show a marked increase in fully convective cooling during the convective period and a small decrease in vapor blanket duration, as found in the cooling curve analysis. This result confirms the accuracy of the created CFD model, and it serves as a powerful tool for evaluating operational parameters, which can be utilized to improve quenching techniques further. Data for the model was drawn from practices employed in companies across numerous industries.

9.1. Future Work

Future research can continue with numerical simulations and a wider range of parameters, including different agitation velocities, the properties of the quenching fluid and other types of boundary conditions to investigate and better understand the dynamics of the quenching process. Exploring material compositions that exhibit different thermal and mechanical properties could also shed light on the versatility of the quenching methods for diverse applications. Furthermore, such studies should also be extended to complex geometries concerning multi-component assemblies, increasing the findings' applicability in real-life industrial settings. Empowering the quenching with state-of-the-art computational tools, like utilising machine learning for predictive modelling and optimization, could also lead to further enhancement and efficiency in the process. The directions listed in this section are intended as an addition to the findings of the present work, which will hopefully help in the evolution of novel yet thoroughly grounded approaches to heat treatment.

References

[1] G.E. Totten, J.L. Dossett, and N.I. Kobasko, *Quenching of Steel*, Steel Heat Treating Fundamentals and Processes, ASM International, pp. 91-157, 2013. [CrossRef] [Google Scholar] [Publisher Link]

[2] Yongxiang Yang, Reinier A. De Jong, and Markus A. Reuter, "Use of CFD To Predict The Performance Of A Heat Treatment Furnace," *Fourth International Conference on CFD in the Oil and Gas, Metallurgical & Process Industries*, Trondheim, Norway, pp. 1-9, 2005. [Google Scholar] [Publisher Link]

[3] Nima Bohlooli Arkhazloo et al., "Experimental and Unsteady CFD Analyses of the Heating Process of Large Size Forgings in a Gas-Fired Furnace," *Case Studies in Thermal Engineering*, vol. 14, pp. 1-12, 2019. [CrossRef] [Google Scholar] [Publisher Link]

[4] Run-Hua Song et al., "An Experimental and Numerical Study of Quenching-Induced Residual Stresses Under the Effect of Dynamic Strain Aging in an IN718 Superalloy Disc," *Journal of Engineering Materials and Technology*, vol. 144, no. 1, pp. 1-13, 2022. [CrossRef] [Google Scholar] [Publisher Link]

[5] Manuel de J. Barrena-Rodriguez, Francisco A. Acosta-Gonzalez, and Maria M. Tellez-Rosas, "A Review of the Boiling Curve with Reference to Steel Quenching," *Metals*, vol. 11, no. 6, pp. 1-17, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[6] Yongxiang Yang, Reinier A. De Jong, and Markus A. Reuter, "CFD Prediction for the Performance of a Heat Treatment Furnace," *Progress in Computational Fluid Dynamics, an International Journal*, vol. 7, no. 2-4, pp. 209-218, 2007. [CrossRef] [Google Scholar] [Publisher Link]

[7] James Jan, and D. Scott MacKenzie, "On the Development of Computational Fluid Dynamics Quenching Simulation Methodology for Effective Thermal Residual Stress Control," *Journal of Materials Engineering and Performance*, vol. 33, pp. 3986-4010, 2024. [CrossRef] [Google Scholar] [Publisher Link]

[8] Amine Kacem, Brahim Bourouga, and Ahmed Ould El Moctar, "Effect of Bath Agitation on the Transient Cooling by Quenching Process," *ASME Journal of Heat Transfer*, vol. 143, no. 12, pp. 1-12, 2021. [CrossRef] [Google Scholar] [Publisher Link]

[9] Kazuaki Kobayashi, Osamu Nakamura, and Yoichi Haraguch, "*Water Quenching CFD (Computational Fluid Dynamics) Simulation with Cylindrical Impinging Jets*," Nippon Steel & Sumitomo Metal Technical Report, Process Technology Div., Process Research Laboratories, no. 111, pp. 101-106, 2016. [Google Scholar] [Publisher Link]

[10] D.S. MacKenzie, "Application of Computational Flid Dynamics to Understand Quenching Problems," *International Federation for Heat Treatment and Surface Engineering 18th Congress*, pp. 5375-5387, 2010. [CrossRef] [Google Scholar] [Publisher Link]

[11] Vedanth Srinivasan et al., "Numerical Simulation of Immersion Quenching Process of an Engine Cylinder Head," *Applied Mathematical Modelling*, vol. 34, no. 8, pp. 2111-2128, 2010. [CrossRef] [Google Scholar] [Publisher Link]

[12] Caner Simsir, and C. Hakan Gur, "A Simulation of the Quenching Process for Predicting Temperature, Microstructure and Residual Stresses," *Journal of Mechanical Engineering*, vol. 56, no. 2, pp. 93-103, 2010. [Google Scholar] [Publisher Link]

[13] Maciej Balcerzak et al., "An Investigation into Sheet-Inconel 718 Forming with Flexible and Metal Tools-Simulation and Experiment," *Materials*, vol. 17, no. 13, pp. 1-24, 2024. [CrossRef] [Google Scholar] [Publisher Link]

[14] Lauralice de Campos Franceschini Canale, and George E. Totten, "Quenching Technology: A Selected Overview of the Current State-of-the-Art," *Materials Research*, vol. 8, no. 4, pp. 461-467, 2005. [CrossRef] [Google Scholar] [Publisher Link]

[15] Dae-Hoon Ko et al., "Application of QFA Coupled with CFD Analysis to Predict the Hardness of T6 Heat Treated Al6061 Cylinder," *Journal of Mechanical Science and Technology*, vol. 27, pp. 2839-2844, 2013. [CrossRef] [Google Scholar] [Publisher Link]

[16] Xia-Wei Yang et al., "Optimum Design of Flow Distribution in Quenching Tank for Heat Treatment of A357 Aluminum Alloy Large Complicated Thin-Wall Workpieces by CFD Simulation and ANN Approach," *Transactions of Nonferrous Metals Society of China*, vol. 23, no. 5, pp. 1442-1451, 2013. [CrossRef] [Google Scholar] [Publisher Link]

[17] ASTM D6200-01, "*Standard Test Method for Determination of Cooling Characteristics of Quenching Oils by Cooling Curve Analysis*," ASTM International, 2022. [Publisher Link]

[18] Edwan Anderson Ariza et al., "Numerical Simulation with thorough Experimental Validation to Predict the Build-up of Residual Stresses during Quenching of Carbon and Low-Alloy Steels," *ISIJ International*, vol. 54, no. 6, pp. 1396-1405, 2014. [CrossRef] [Google Scholar] [Publisher Link]

[19] David D. Hall, and Issam Mudawar, "Experimental and Numerical Study of Quenching Complex-Shaped Metallic Alloys with Multiple, Overlapping Sprays," *International Journal of Heat and Mass Transfer*, vol. 38, no. 7, pp. 1201-1216, 1995. [CrossRef] [Google Scholar] [Publisher Link]

[20] Peter Fernandes, and K. Narayan Prabhu, "Effect of Section Size and Agitation on Heat Transfer during Quenching of AISI 1040 Steel," *Journal of Materials Processing Technology*, vol. 183, no. 1, pp. 1-5, 2007. [CrossRef] [Google Scholar] [Publisher Link]