CFD simulation of the multiphase heat transfer during the quenching process

J Novosád1* P Peukert1 N Pomp1 P Klouček1
1VÚTS, a.s., Svárovská 619, Liberec, Czech Republic

*jan.novosad@vuts.cz

Abstract. The paper presents the results of the CFD (Computational Fluid Dynamics) simulation of quenching process for the ring shape sample. The aim of the work is to develop and validate the methodology for multiphase CFD simulation including the boiling during the quenching process. CFD simulation is provided in ANSYS Fluent. The Lee model is used for modelling the phase change during the quenching process. The first step consider the simulation of cooling of the ring sample when the correct model parameters will be found. Validation of results is performed by comparison with experimental data. Experimental was realized inside the own designed quenching bath device filled with quenching polymer. The general description of the experimental setup is included in the paper. The CFD results are cooling curves, i.e. variation of solid temperature on time. The Lee model parameters especially the evaporation frequency was tuned. Thin polymer film on the solid surface was considered to bring the results closer to the experimental data. The comparison between experiment and CFD shows very good agreement for higher temperatures, which covers the boiling stage. On the other hand for lower temperatures worse match of results was found caused probably by the sensitivity on the inlet velocity profile settings. Some recommendations for future work were defined.

1 Introduction

In mechanical engineering, the parts production process contains a heat treatment and quenching used to improve physical and chemical properties of a material, e.g. strength, toughness, durability etc. The quenching process uses the quenchant as a medium for controlled cooling of the parts.

The heat transfer [1] during quenching gradually passes three stages [2] (illustrated in figure 1):

- Vapor film stage.
- Boiling stage.
- Convection stage.

Vapor film stage is represented by vapor barrier formation on the surface of the part. The barrier is cause by the thin vapor film with low thermal conductivity, i.e. low heat transfer rate.

Boiling stage begins by the vapor film collapse. Then the nucleate boiling, characterized by high heat transfer rate, continues until the temperature of the surface falls below the boiling point of the liquid. Convection stage, i.e. convection heat transfer, is the final stage of the quenching process. The heat transfer rate is the slowest.

As the quenchant different kind of liquid, solid or gas could be used. In our case the polymer quenchant is used as the solution of the organic polymer in water. The using of polymer gives some advantages [2] comparing to using of oil quenchants. The quenching speed could be maintained by varying the concentration. The uniform polymer film avoids the soft spots and reduces surface
thermal gradients and residual stresses caused by non-uniformity in heat transfer. Finally there is no smoke and fume occurred during quenching which improves the environment quality.

![Figure 1. Quenching stages](image)

The experimental investigation of the thermal field inside the quenched part during the industrial process could be very difficult. However, many authors deal with the investigation of quenching process using the CFD (Computational Fluid Dynamics [3]) analysis. In paper [4] MacKenzie have described the abilities of CFD in ‘real world’ applications of quenching. Then some design study about affecting the flow of quenchant were performed, e.g. by Kobayashi et al. [5]. The main advantage of modelling is the coupling between CFD and FEA (Finite Element Analysis) analysis, described e.g. by Banka et al. [6]. These coupled studies can give as the results not only the fluid and thermal fields, in addition the chemical and mechanical parameters, i.e. the austenite decomposition, internal stress state and distortion of the finished part.

The aim of our study is to develop and validate the methodology for CFD analysis. For phase change the Lee [7] model will be used. The validation of the CFD results will be based on the experimental results. The measurement will be provided inside the quenching tank with our design.

2 Experimental part
First of all the experimental quenching device (see figure 2) was designed and built in VÚTS. The main part is the welded steel tank with the filling volume approx. 1.4 m³. The circulation of polymer PVP (polyvinylpyrrolidion) solution is performed by two pumps on the sides of the tank. In the bottom of the tank there are bent sheets for directing the flow. The main flow inside the tank is vertical from bottom to top. Quenched parts are situated on the grid which is submerged in the tank.

![Figure 2. Quenching tank](image)
2.1 Samples for experiment
For the experiment two samples with different shapes (shown in figure 3) were prepared. Thermocouples (TC) for temperature measurement were installed in both samples. Positions of TCs were chosen so as not to affect the flow and to prevent damage of the thermocouples during measurement. Positions are illustrated in figure 3 (red crosses). Samples prepared for measurement, including mounting of thermocouples, is shown in figure 4.

![Figure 3. Sketch of samples – a) ring, b) bell-shaped.](image)

![Figure 4. Samples with thermocouples.](image)

2.2 Setup and preparation
Temperature measurement was performed using the data logger Ahlborn ALMEMO MA5690-2 with two thermocouples Ahlborn FTA05L0500 for measuring temperature of samples. In addition, two thermocouples were connected to measure temperature of the quenching bath and inside the furnace for heating up the samples. Temperatures were measured throughout the whole process with the sample rate 1 Hz.

At first both samples were heated up to 850 °C and maintained at that temperature for several tens of minutes (30 min for ring, 90 min for bell-shape). Then the first sample was putted on the grid and submerged into the quenching bath maintained at 30 °C. The manipulation time does not exceed 15 s. The bath was mixed by pumps. Quenching was stopped when the sample temperature reaches the value under 50 °C. After that the whole process was repeated for the second sample.
3 CFD analysis

The main goal of our work was to develop the methodology for CFD simulation of the quenching process and validate that by the experimental results. The simulation have to be based on the same conditions as for experiments, i.e. geometry and initial temperature of samples, temperature of bath etc. For the first stage of simulations the ring shape sample data will be used.

3.1 Geometry and mesh

Based on the geometry of samples (see figure 3) the CAD model of the ring was created respecting the dimensions. On the other hand some elements were neglected for simplification, e.g. screws and holes for thermocouples. Then symmetry was used to create final 2D axisymmetric CAD model. Geometry for simulation is only the cutout around the ring, not the whole quenching bath.

Computational grid is composed as two domains, one for fluid, second one for solid. The quad grid with several triangles was created. The grid was refined near the interface. Total number of elements is approx. 10 000 for ring.

3.2 CFD model

The simulated problem could be considered as the transient multiphase flow with boiling on the hot wall. The mixture model was used for multiphase flow of two fluids, i.e. polymer solution and its vapor. The Lee model [8] was used for evaporation (boiling). The influence of the Lee model parameters, i.e. tune the evaporation and condensation frequency is a part of our work.

The properties of fluids were temperature dependent. The values of density, thermal conductivity and viscosity were used from the company material database respecting the solution concentration.

The calculation is transient with variable time step respecting the flow Courant number Co < 2. Final time is the same as was in the experiment, approx. 300 s for ring.

3.3 Boundary and initial conditions

The boundaries of the model are consistent with the experiment. For simplification the symmetry of the models was applied. Boundary conditions of the models are illustrated in figure 5. Symmetry is applied to create the axisymmetric model. At the inlet the velocity was given as 0.4 m/s in accordance with the previous CFD study of quenching bath. Moving wall is applied to simulate the far field, hence the velocity value and direction is the same as for inlet. The volume fraction of vapor is 0 at the inlet. Outlet is set as pressure outlet boundary.

At the beginning the initial temperature of solid is set to 850 °C. Then during the calculation the temperature at point P (red crosses in figure 5) is reported for future evaluation per each 0.1 s.

![Figure 5. BCs of the computational domain – ring.](image)
4 Results
The results show the comparison of temperature versus time from the simulation of the ring with different settings of flow parameters. Several calculations were provided to tune up the Lee model parameters. The best fitted parameters were used for the final calculation.

4.1 Ring
The cooling curves obtained from the experiment and three CFD cases are shown in figure 6. The first CFD case describes the quenching by natural convection, i.e. the inlet velocity is set as 0. The flow is driven by buoyant forces. During the boiling stage the flow is speed up by the water vapor. It is clear that the vapor film slows down cooling. In comparison with the experimental data less slope of the curve shows slower cooling than during the experiment.

The inlet velocity in the second case was set to value 0.4 m/s. Thus represents the nominal value in the quenching tank. The good agreement is observed for temperatures above 500 °C. For lower temperatures the curve extends below experimental data that means the cooling is faster. These two CFD cases results were used for the tuning of the coefficients and „more real“ boundary conditions for the final case.

The final case was modified in several ways. The inlet velocity was reduced to only 0.3 m/s. This should reduce the non-reality of the uniform velocity profile at the inlet of the model. The second modification covers the boiling process when the thin polymer film was defined on the solid surface. Finally the Lee model parameters were tuned to minimize the difference between CFD and experimental results during the boiling stage. It could be said that the good agreement is obtained for the temperatures higher than 300 °C. For lower temperatures the velocity field was probably inhomogeneous during the experiment therefore the velocity apparently was not constant. This could lead to decreasing of the cooling speed.
5 Conclusion

The aim of our work was to develop and validate the methodology for CFD simulation of the quenching process. Several cases were solved to set up and tune the multiphase flow model including evaporation. The experimental data was used for validation. The comparison with CFD data for particular cases shows (see figure 6), that the inlet velocity value significantly affects the results. The Lee model parameters especially the evaporation frequency was tuned. The inclusion of a thin polymer film on the solid surface was also necessary to bring the results closer to the experimental data.

To conclude, the comparison between experiment and CFD shows very good agreement for higher temperatures, which covers the boiling stage. At this stage the main influence on cooling is by heat transfer caused by evaporation, i.e. the correctness of the results depends on the parameters of the boiling model. This part was fully done which is confirmed by the match of results. On the other hand the convection stage is more affected by the fluid flow. In this part worse match of results is probably caused by the insufficiently accurate inlet velocity specification because no experimental data was available.

Finally, the measured flow field data as the input for CFD could improve the quality of results for the convection stage. This should be done before the CFD methodology will be tested for the bell shape sample. In the future it is assumed to use the methodology for the industrial cases.

Acknowledgments

This publication was supported by the Ministry of Industry and Trade (MPO) within the framework of institutional support for long-term strategic development research organization - provider MPO, recipient VÚTS, a. s.

References

[1] Lienhard J H IV, Lienhard J H V 2018 A heat transfer textbook, 4th Ed. (Cambridge Massachusetts: Phlogiston Press)
[2] MacKenzie D S 2003 Advances in quenching Houghton International, Inc.
[3] Ferziger J H, Peric M 2002 Computational methods for fluid dynamics (Berlin: Springer-Verlag) ISBN 3-540-42074-6
[4] MacKenzie D S 2010 Application of computational fluid dynamics to understand quenching problems 18th Congress International Federation for Heat Treatment and Surface Engineering
[5] Kobayashi K, Nakamura O and Haragucchi Y 2016 Water quenching CFD (Computational Fluid Dynamics) simulation with cylindrical impinging jets Nippon steel & sumitomo metal technical report 111
[6] Banka A, Franklin J, Li Z, Ferguson L B and Aronov M 2008 CFD and FEA used to improve the quenching process Heat treating progress September 2008 pp 50-56
[7] Lee W H 1979 A Pressure Iteration Scheme for Two-Phase Modeling (Los Alamos: Scientific Laboratory Report) LA-UR 79-975
[8] Ansys Fluent theory guide Ansys, Inc.