ACCURATE MODELING OF QUENCHING PROCESSES USING CFD AND A FLOW BOILING DATABASE

Experimentally derived surface heat flux rate data and CFD software are helping researchers more accurately simulate liquid quenching processes.

Andrew L. Banka* and Jeffrey D. Franklin,* Airflow Sciences Corp., Livonia, Mich.

Computational analysis, a tool that has helped optimize many heat treating processes, has yet to have an extensive impact on liquid quenching. Quenching in a vaporizable liquid is a critical production step for many metal parts, but the complexity of the physics make it a difficult process to simulate. Rigorous physics-based boiling models can yield good results, but are impractical for cases of industrial interest. An alternative, data-driven approach presented here holds promise, however, and is scoring well when tested under realistic conditions.

A MODELING DILEMMA

Heat treat processes incorporating a liquid or gas quenching step are essential to the quality of manufactured metal components. The rapid cooling that occurs during quenching largely determines phase distribution, microstructure, residual stress, and distortion in the as-quenched part. Manufacturers, naturally, would like to better control these pivotal material properties, and are looking to numerical analysis tools for the necessary leverage.

Numerical methods and tools have been used to gain a better understanding of many heat treating operations, leading to significant process improvements. But their use in the area of liquid quenching has been much less extensive. Simulating the process of quenching has been a persistent challenge because of the difficulty in predicting surface heat flux rates due to boiling of the vaporizable quenchant.

Materials scientists confronting this issue will often conduct instrumented tests on the part in question, then use "inverse analyses" to extract surface heat flux values from the measured internal temperatures. This approach is time consuming, expensive, and subject to non-unique solutions. Further, the resulting heat flux values are only applicable to the specific part and quenching environment. If a change will be made in processing conditions, the experiment must be repeated.

A purely computational approach—one that predicts the explosive formation of vapor and release of bubbles based on the underlying physics—is theoretically possible, but it would be impractical in applications of industrial interest because length scales and time frames differ by orders of magnitude across the working domains.

An alternative approach, and the focus of this article, incorporates both experimental and computational methods, benefiting from the combination of strengths. Elusive surface heat flux rates are determined experimentally in a precisely controlled flow boiling test system. This data is then used to build computational fluid dynamics (CFD) models that are proving to be quite accurate in quenching process simulations.

FLOW BOILING DATA COLLECTION

As part of an Air Force Research Laboratory sponsored SBIR study, a group of investigators developed an experimental facility to collect flow boiling heat flux data for vaporizable quenchants. In the heart of the system, as shown in Fig. 1, a test coupon embedded in the wall of a flow channel is exposed to quenchant circulating at controlled speeds. The heat source, a bank of cartridge heaters with an output

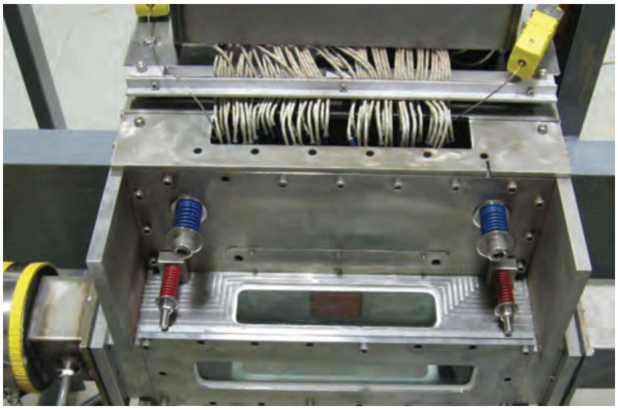

Fig. 1 — Flow channel for measuring flow boiling surface heat flux rates.

---

*Member of ASM International

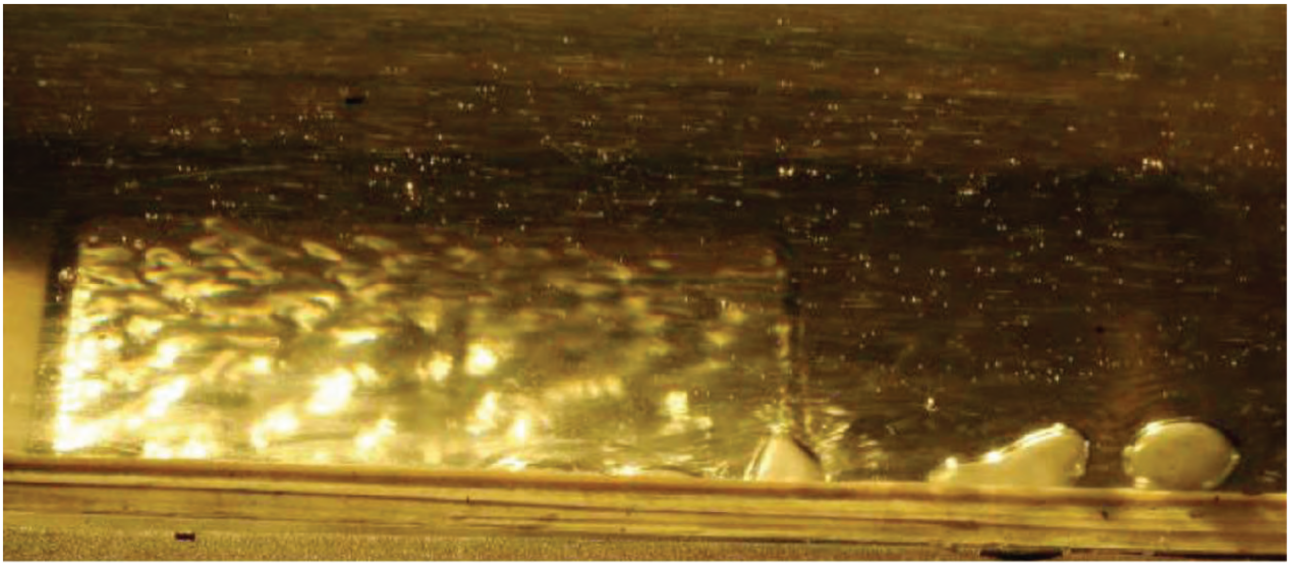

Fig. 2 — Film boiling on side surface at low velocity and high temperature. Flow is from left to right. Vapor film is thick and uneven, and vapor bubbles are being shed.

of 27 kW, is capable of heating the coupon surface to 1600°F, achieving a maximum surface heat flux of 10.8 MW/m^2.

Data collection begins when the coupon is at its maximum temperature and stable film boiling is established at the desired quenchant flow rate and temperature. Collection continues as the heaters reduce the surface temperature of the coupon at a controlled rate of 20°F/minute. This slow cooling approach provides a well-defined relationship between surface temperature and surface heat flux during pseudo-steady-state conditions. Figure 2 shows the vapor-liquid interface on the test coupon during film boiling at a relatively low flow rate. The test procedure is then repeated for each flow velocity, quenchant temperature, and surface orientation of interest.

The raw data collected are temperature values obtained from thermocouples inserted below the surface of the copper coupon. The values are used to derive surface temperature and heat flux rate. The procedure may seem similar to the inversing processes typically used in quench trials, although there are a few key differences. First, all data are collected in near steady-state conditions, eliminating the complexities inherent in transient cooling processes. Second, two thermocouples are used at each surface location, placed at different depths, allowing for advanced extrapolation techniques to the surface of the coupon[1]. And finally, because surface heat flux data are linked to surface temperature, quenchant temperature, quenchant velocity, and surface orientation, they can be applied to any surface under the same conditions.

QUENCH TESTING, SIMULATION AND VALIDATION

To validate the method, quenching data were collected on a generic turbine disk shape made of Inconel 718. Nearly two dozen thermocouples were inserted into the part at critical locations. Most of the thermocouples were placed 0.100 in. from the surface, while five were buried deeper into the part. Near the rim of the disk, the near-surface thermocouple was repeated at 60° intervals to assess the circumferential uniformity of the quenching operation.

In quench tests performed at the Wyman-Gordon research facility in Houston, the disk was soaked at 1800°F, then quenched in Houghton 3420 quenchant under both still and agitated conditions. Temperature data were collected at 10 Hz during both the heating and quenching cycles.

The quenching operation was simulated with ANSYS-Fluent version 16, using a 0.1 second time step. For the still oil quenching simulation, radial symmetry of the part allowed for a 2D model, and 17,557 cells were used to represent both the disk and quench tank in cross section. Simulating a quench of 1500 seconds required about five hours of processing time on a 3.5-GHz Linux workstation with eight processors.

Boiling functions constructed from the flow boiling database were incorporated into a Fluent User Defined Function (UDF) that adjusted the surface heat transfer coefficients (HTCs) on an iterative basis for each surface

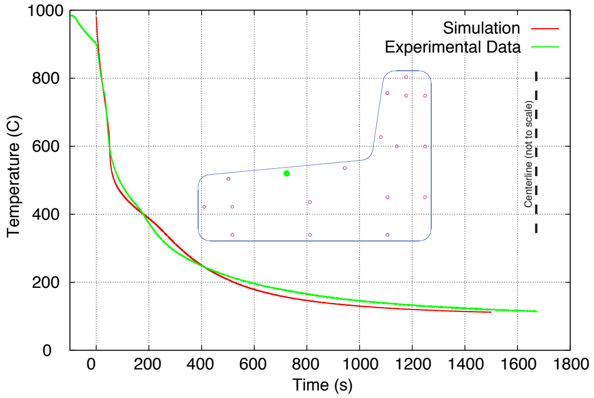

Fig. 3 — Location with best correlation for still oil generic turbine disk.

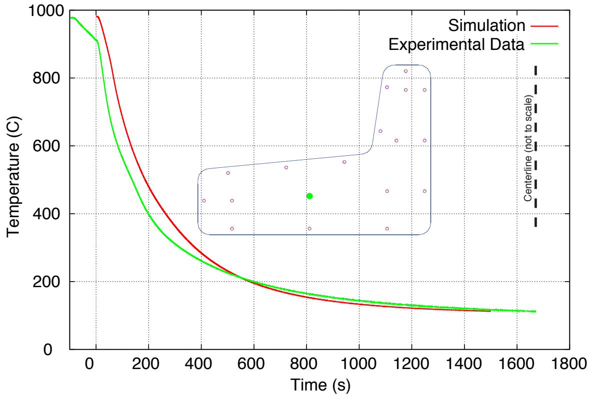

Fig. 4 — Location with poorest correlation for still oil generic turbine disk.

face within the computational domain. Each computational face is thus treated as a separate heat transfer zone with the HTCs based on local conditions. Temperature-dependent properties were used for both Inconel 718 and Houghton 3420.

Temperature data were recorded during the simulation process at each thermocouple location to compare against experimentally measured values. The degree of correlation at each location was based on the average difference between the simulated and measured temperatures over 1500 seconds of simulated time. This is essentially the integrated area between the simulated and measured cooling curves divided by total simulation time.

For the still oil case, average temperature deviations ranged from 14.9° to 28.5°C, indicating that the simulation provides a good representation of the actual quenching behavior. For the thermocouples on the rim of the part, the average spread of temperature data was 27.3°C, while the average deviation between simulated temperature and the mean of the six rim thermocouples was just 22.2°C. In other words, the correlation of the simulation was within the variation band for the actual process.

Cooling curves for locations with the least and greatest average deviations are shown in Figs. 3 and 4. The curves in Fig. 3 are in close agreement over the entire cooling range. The agreement in Fig. 4 is not as good, but it still predicts overall cooling behavior fairly well. The location of the greatest deviation in a buried thermocouple is curious, and may suggest a difference between simulated and actual thermal properties of Inconel 718.

CONCLUSION

A method has been developed combining carefully collected flow boiling heat flux data with CFD simulations to provide accurate simulations of quenching operations. A comparison of simulated and experimentally measured cooling rates for a non-trivial geometry has shown good correlation, indicating that this tool provides a practical method of assessing and improving industrial quenching operations. The level of correlation shown in the example indicates that the experimental database is not limited to the original test conditions (as with typical quench trials), but rather has broad applicability. **~HTPro**

Acknowledgments

This material is based on work supported by the United States Air Force under Contract No. FA8650-12-C-5110. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the author(s) and do not necessarily reflect the views of the United States Air Force. Airflow Sciences would also like to thank members of the Air Force Research Laboratories Nickel Residual Stress FEP program (United States Air Force Contract No. FA8650-13-2-5201) for their collaboration on work related to the generic turbine disk shape.

Reference

1. Edward B. Coy, Measurement of Transient Heat Flux and Surface Temperature Using Embedded Temperature Sensors, *Journal of Thermophysics and Heat Transfer,* Vol 24, No. 1, 2010.

For more information: Andrew L. Banka is technical director of Airflow Sciences Corp., 12190 Hubbard St., Livonia, MI, 48150, 734.525.0300, abanka@airflowsciences.com, www.airflowsciences.com.