Research Programs
GAS Quenching – Understanding, Controlling and Optimizing the Process – I

Research Team:

M. Maniruzzaman
Jiankun Yuan
Prof. Richard. D. Sisson Jr.

Performance, safety and environmental considerations can make gas quenching an attractive alternative to conventional oil, polymer and water quenching. But up to now, it has not experienced wide spread implementation due to some issues restricting introduction of this technology. Compared with liquid quenching, the specific advantages of gas quenching include cleaner product surfaces, avoidance of need for washing the parts after quenching, and minimization of environmental problems associated with the handling and disposal of liquid quenching fluids, all these lead to lower the operating costs of a hardening process.

Due to poor thermal transfer characteristics (lower cooling rate) of gases under normal conditions, they have to be optimized by proper adjustment of gas pressure and flow speed. In principle, gas quenching can be performed in two ways, namely: at low or atmosphere pressure with high gas velocity or at high pressure with limited gas velocities. For the second technique, the vacuum furnace is a natural choice.

The objectives of this project are --

- To develop a theoretical and experimental understanding of the process variables that controls the high-pressure quenching process. In addition, to determine the capability of gas quenching as compared to quenching in other media.

- The “key” issues in the design, analysis, characterization and operation of a gas quenching system will be identified and investigated. These issues include:
    - The accurate prediction of the heat transfer coefficients on the quenched part surfaces as a function of:
        - Metal surface temperature
        - Gas composition and pressure
        - Agitation
        - Quench vessel geometry
    - The prediction of the gas quenched part’s microstructure and deformation.

To achieve these objectives the following strategy are taken:

- The theoretical and modeling work will be completed at WPI using Computational Fluid Dynamics (CFD) software and fundamental heat transfer analyses. In addition, the DANTE, an add-on module of ABAQUS Finite Element (FE) Software will be evaluated for heat treatment related stress / distortion and microstructure predictions. Computer predicted results will be compared with experimental results.

- Experimental work will be conducted at CHTE member companies in commercial facilities. Quenching after vacuum carburizing and inert gas heat treating will be tested. The temperature versus time data will be recorded for simple quench probes and more complex part geometries and orientations. The data will be analyzed to determine the heat transfer coefficients as a function of gas composition and pressure as well as gas velocity / agitation for several alloys and part geometries and orientations. These results will be compared with the predictions based on the fundamentals of heat transfer and computational fluid dynamics models.

CFD method is mainly used to study the interactions between the quenching gas and quenched parts in the quench chamber. Major task of CFD is to provide the local convective heat transfer coefficient as a function of temperature on the surface of parts, and then the heat transfer coefficient can be used as one of the input parameters for FE analysis of material response of quenched parts. Velocity and pressure distribution in furnace provided from CFD modeling can also be the important factors to analyze non-uniformity of heat transfer effects frequently occurred in past heat treatment history. With CFD technique, more systematic investigation can be easily conducted to study the effects of circulating fan speed, utilization of different quenching gases and pressure variation on heat transfer effects in quenching process. Combined with FE analysis, CFD can provide beneficial suggestions for modification of existing furnace or arrangement of quenched parts in chamber, and guidance for new developed gas quenching systems.

Fig. 1 shows the CFD result of a study of flow field in a vacuum furnace [1]. The non-uniform flow field caused distortion of the quenched part in the charge zone of the furnace. Suggestions from CFD results were used to modify the design and alignment of parts and furnace structure. One of the important findings of the CFD study was the recirculation eddies in the charge zone.

To avoid the recirculation, a perforated plate was utilized at inlet of charge zone to straighten the flow.

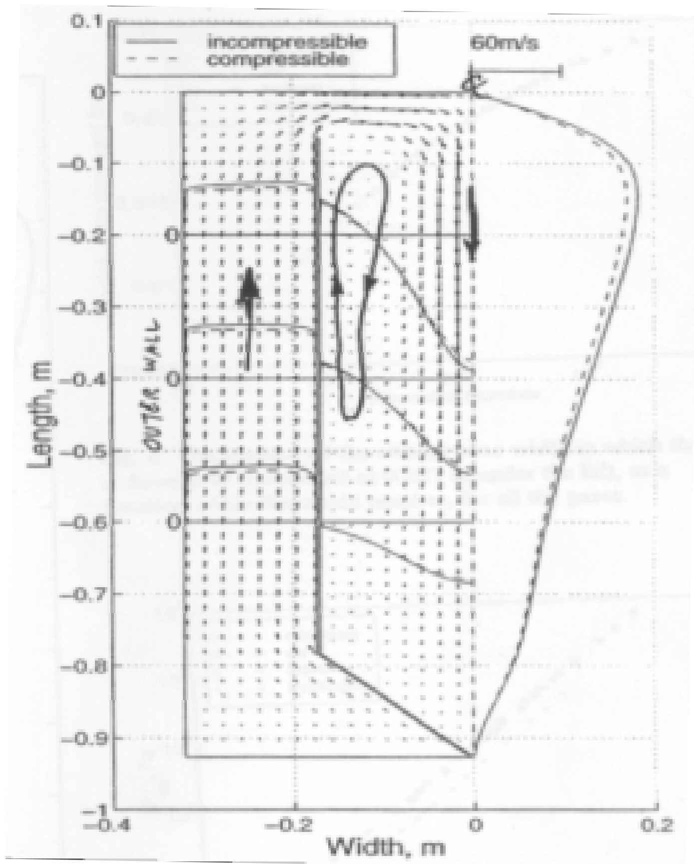

Fig.1 Flow field distribution in the chamber obtained from CFD simulation, for flow of nitrogen at 50bar, 20 ^{0}C, and average of velocity of 30m/s in the charge zone. The fastest down flow in the duct is at the centerline, a recirculation vortex can be seen next to the charge zone wall. Comparing results for gas assumed as compressible and incompressible. Only one half is shown due to symmetry [1].

FE method is a powerful and effective tool for structural analysis. Recently this method has been used successfully for structural prediction during heat treatment. In the past thirty years, general-purpose commercial finite element programs have been widely developed and used in industry and are becoming more popular for engineering analysis and equipment design. Some FE programs have been used as backbone for development of some specific software for industrial applications, one of such example is DANTE developed on ABAQUS for metal heat treatment. Fig. 2 is an example of using DANTE to investigate the deformation and residual stress of a coil spring under oil quenching and intensive quenching (IQ) [2].

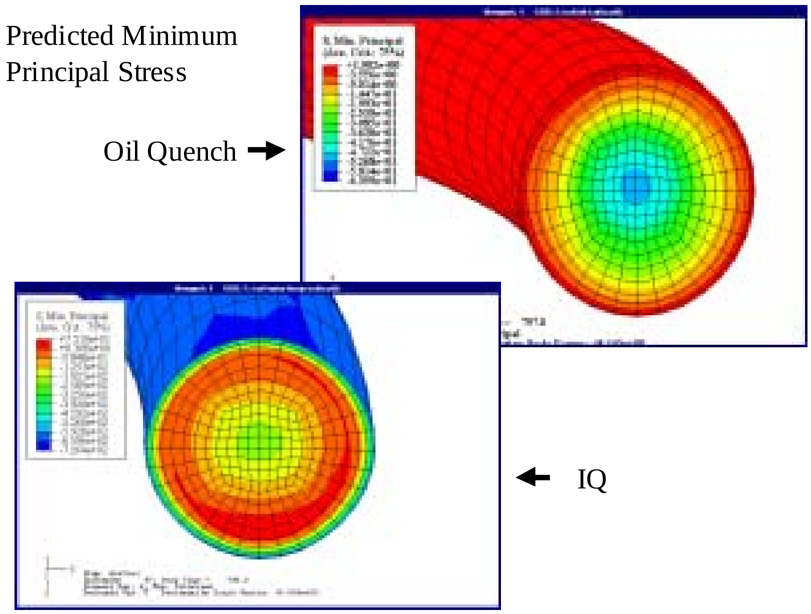

Fig. 2 Comparison of a coil spring residual stress resulted from oil and intensive quenching, based on DANTE simulation [2]

References

1. J. Ferrari, N. Ipek, N. Lior and T. Holm, *Flow consideration in quenching vessel. Proc. 3rd ASM Int'l Conf. On Quenching and Control of Distortion, Prague, Czech Republic*, 1993.
2. B. Lynn Ferguson & Andrew M. Freborg, *How Does Intensive Quenching Produce High Surface Compressive Stress: Understanding the Process through Computer Simulation*. The 13th International Federation for Heat Treatment & Surface Engineering Conference. October 8, 2002.