13th Australasian Fluid Mechanics Conference
Monash University, Melbourne, Australia
13-18 December 1998

Fluid Flow During Gas Quenching of Steel Components

Ihab ElKatatny, Yos Morsi, Aaron Blicblau and Derry Doyle

School of Engineering and Science
Swinburne University of Technology, Victoria 3122, Australia

ABSTRACT

The gas flow and heat transfer phenomena occurring inside a vacuum furnace, in which the gas quenching process is carried out, have been studied using the computational fluid dynamics package CFX. Currently optimisation of the system is being conducted. It is evident from the simulation that there is some loss of gas quench efficiency due to the development of gas recirculation areas within the hot zone which, in turn, influences the temperature profiles generated in the load. The configuration of the load with respect to the gas nozzles in quenching has been investigated. With respect to the large die block modelled in this study, oscillating the gas nozzles and alternating the gas quenching from the top and bottom of the hot zone were found to have surprisingly little influence on thermal gradients within the load. A reasonably close agreement was obtained for cooling curves determined experimentally for marquenching with the predicted results.

INTRODUCTION

Whilst heat treatment often represents less than ten percent of the finished cost of a metal part, the correct application of heat treatment to the metal can make the difference between success and failure in providing the required performance. Furthermore, its correct application can provide an effective competitive edge in the market (Cavallaro et al, 1993). Thus, there is a potential for the development of methods which simulate the heat treatment cycle leading to an optimization of procedures to achieve desired geometric and metallurgical properties at minimum costs. In particular the use of computer simulation procedures will substantially reduce the overall heat treatment cost whilst providing easy access to all scales of manufacturing. At the present time. there have been many heat transfer simulation procedures developed which may predict the temperature distributions of both the furnace and the metallic components (Rodic and Rodic, 1983, Arola et al, 1992)). However, due to mathematical complexities and large requirements of computer time, these have been developed for simple geometric shapes utilized in the aerospace industry (Shapiro, 1984) The simulation of complex geometric shapes has proved to be more difficult to model due to the complexity of advancements in mathematical procedures required.

The present study extends previous work(Chen, 1994) through the development of simulation models for fluid flow and heat treatment procedures. It will enhance the body of knowledge in a rapidly advancing field. The practical significance of the project lies in its potential to advance the final stages of manufacturing metallic components. Medium and small scale manufacturers will be able to predict final properties without the need to resort to expensive trials. These models can be adapted to a wide variety of process routes and offer the potential to improve the efficiency of operations. This will in turn provide them with a distinct marketing advantage over particularly overseas manufacturers employing traditional methods.

The present approach links the simulation of heat treatment and final shape control with metallurgical developments by the use of Computational Fluid Dynamic procedures and Finite Element Analysis together with specifically written subroutines (Chen et al 1993). It is an advance on current procedures as it will enable the use of a single model to predict three phases of the heat treatment process, i.e. the heat transfer in the component, the resulting thermal stresses, and the final shape. It is now well accepted that the current metal manufacturing industries have an immediate need for such models.

SIMULATION MODEL

Mathematical model. The mathematical equations for gas flow and heat transfer used by *CFX-F3D*, for high Reynolds number flow, can be described as follows;

- Continuity equation $$\frac{\partial \rho}{\partial t} + \nabla.\left(\rho U\right) = 0 \qquad (1)$$
- Momentum equation

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot \left(\rho U.U\right) - \nabla \cdot \left(\mu_{eff}\ \nabla U\right) = - \nabla p + \nabla \cdot \left(\mu_{eff} \left(\nabla U\right)^T\right) + B \qquad (2)$$

Where ρ and U are mean fluid density and velocity, *p* the pressure, *t* the time, *B* body force and μ_{eff} is the effective viscosity defined by

$$\mu_{eff} = \mu + \mu_T$$

where μ_T is the turbulent viscosity defined by

$$\mu_T = C_\mu \rho \frac{k^2}{\varepsilon}$$

- Energy equation

$$\frac{\partial \rho H}{\partial t} + \nabla \cdot (\rho U H) - \nabla.(\lambda \nabla T) = \frac{\partial p}{\partial t} \quad (3)$$

Where T is the temperature, λ is the thermal conductivity and H is the enthalpy.
The *CFX-F3D* solve the heat conduction equation for the temperature in the solid region as follows;

$$\frac{\partial (\rho_S H)}{\partial t} - \nabla.(\lambda_S . \nabla T) = 0 \quad (4)$$

where $H = C_S T$, and ρ_S, C_S, λ_S are the density, specific heat, and the thermal conductivity of the solid.
Turbulence model. RNG k-ε model is used as an alternative to the standard *k-ε* model for high Reynolds number flow (4). The equations used to describe the turbulence model are:

$$\frac{\partial \rho k}{\partial t} + \nabla \cdot \left(\rho U k\right) - \nabla \cdot \left(\left(\mu + \frac{\mu_T}{\sigma_k}\right) \nabla k\right) = P + G - \rho \varepsilon \quad (5)$$

and

$$\frac{\partial \rho \varepsilon}{\partial t} + \nabla \cdot \left(\rho U \varepsilon\right) - \nabla \cdot \left(\left(\mu + \frac{\mu_T}{\sigma_\varepsilon}\right) \nabla \varepsilon\right) =$$

$$\left(C_1 - C_{1RNG}\right) \frac{\varepsilon}{k} \left(P + C_3 \max\left(G,0\right)\right) - C_2 \rho \frac{\varepsilon^2}{k} \quad (6)$$

where k is turbulence kinetic energy, ε is turbulence dissipation rate and P is the shear production defined by:

$$P = \mu_{eff} \nabla U.(\nabla U + (\nabla U)^T) - \frac{2}{3} \nabla.U(\mu_{eff} \nabla.U + \rho k) \quad (7)$$

and G is production due to the body force defined by:

$$G = \frac{\mu_{eff}}{\rho \sigma_\rho} g.\nabla \rho \quad (8)$$

C_1, C_2 and C_3 are model constants and $C_{1\,RNG}$ is given through the equation:

$$C_{1RNG} = \frac{\eta\left(1 - \frac{\eta}{\eta_o}\right)}{\left(1 + \beta \eta^3\right)} \quad \text{and} \quad \eta = \left(\frac{P_s}{\mu_T}\right)^{0.5} \frac{k}{\varepsilon}$$

Where η_0 and β are additional model constants, and P_S is the shear part of the production, that is the first term in Eq. 7.

FLOW ANALYSIS

The investigation was carried out under different load configurations and different operating pressures. and the experimental comparison were performed on an H13 steel die whose size was 560mm x 350mm x 350mm. A Schematic diagram showing the furnace together with the hot zone and gas inlet and outlet for quenching in relation to the load configuration is shown in Figures 1a, 1b and 1c respectively, for two possible orientations of the load horizontal and vertical with respect to the gas inlet and outlet points in the hot zone. It should be noted that when the gas enters from the bottom of the hot zone then the gas inlet (nozzles) and outlet are the reverse of that shown in the Figure 1b. Nitrogen, at a pressure of 1 bar and velocity of 50 ms^{-1}, entered the top of the hot zone through twelve nozzles. The computer simulation of a direct quenching process inside the hot zone from an austenitising temperature of 1030 °C was carried out using *CFX (CFX,1995)*. The simulation used twenty five thousand cells to describe the hot zone and load geometry as shown in Figure 2.

RESULTS AND DISCUSSION

In the first stage of quenching for both surface and core temperatures, a close agreement is evident between the numerical simulation and the experimental data. For the second stage of quenching, both the numerical simulation and the experimental data are in reasonable agreement only for the surface temperature. The numerical simulation gives a faster cooling rate for the core when compared to the experimental data.
Figures 3a and 3b show the prediction of gas velocities inside the hot zone for the load in the horizontal configuration. The gas is injected directly into the hot zone at a pressure of 1 bar and an initial velocity of 50 ms^{-1}. The X-Y cross-section shown in Figure 3a indicates that as the gas hits the treated load, it spreads widely toward the side walls of the hot zone creating areas of gas recirculation close to the side surfaces of the treated load. Also, the gas velocity drops sharply from 13 ms^{-1} close to the top surface of the load to around 1 ms^{-1} close to the bottom surface of the load. The gas nozzles operating outside the dimensions of the load direct the quenching gas downwards creating large areas of gas recirculation at the front and back walls of the hot zone as shown in Figure 3b.
Two different regimes of quenching simulations, direct quench and marquench, were investigated. For direct quench, three gas pressure simulations, 1 bar, 4 bar and 6 bar were assumed in order to optimise the cooling time and distortion of the component.
The simulation of the heat treatment cycle for marquenching was as follows: the die was heated by convection at 2 bar nitrogen to a preheat temperature of 870°C and then to the hardening temperature of 1030°C. After adequate soak time nitrogen was admitted to the furnace at 4 bar at a velocity of 60 ms^{-1}. A marquench hold was set at 370°C followed by quenching to room temperature at 2 bar nitrogen.

Numerical Simulation of Marquenching

A three-dimensional computational model was developed to simulate marquenching of the die as detailed in the flow analysis section. In order to reduce the computational time, the simulation was carried out under steady state conditions with the gas entering from the top of the hot zone. The gas inlet velocity was 60ms^{-1} at a quenching pressure of 4 bar. Based on the gas velocity distributions predicted a transient study was carried out to calculate the temperature distribution inside the load at different stages of the quenching process.
Figure 4 shows the comparison between the numerical and experimental data. It can be seen that during the first stage of quenching, the surface temperature records 1.3 minutes to quench to 700°C and 3 minutes to quench to

550°C. The core reached 700°C in 24 minutes and 550°C in 43 minutes. The isothermal hold was set at 370°C with the quench pressure reduced to 2 bar close to the isothermal hold set point. The second stage of quenching starts when the core temperature is within 58°C of the die surface temperature.

Flow Variation

In an attempt to improve the gas velocity distribution it is possible, with the type of vacuum furnace under investigation, to oscillate the gas nozzles(Katatny et al 1997). In the present study, a simulation was carried out with the gas nozzles oscillating through ±30°. The effect of such oscillation was found to have little or no effect on the cooling rate inside the treated load in the vertical configuration as shown in Figure 5.

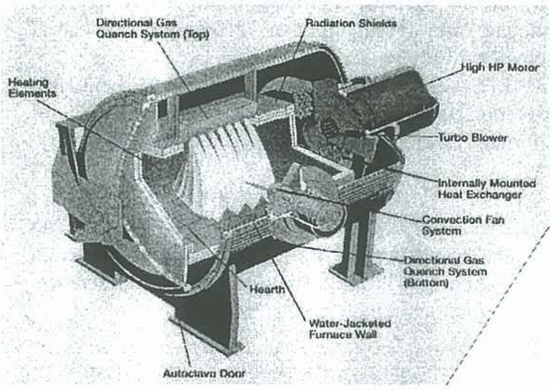
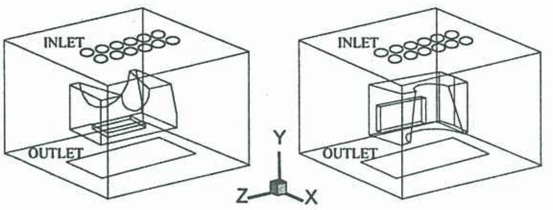


Figure 1a Representation of the furnace showing placement of the nozzles and the die



(b) horizontal (c) vertical

Figure 1B and Figure 1c. Schematic diagram of the hot zone and gas inlet and outlet for quenching in relation to the load configuration.

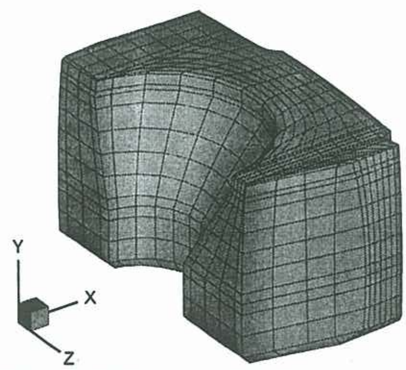


Figure 2. Grid pattern of the die

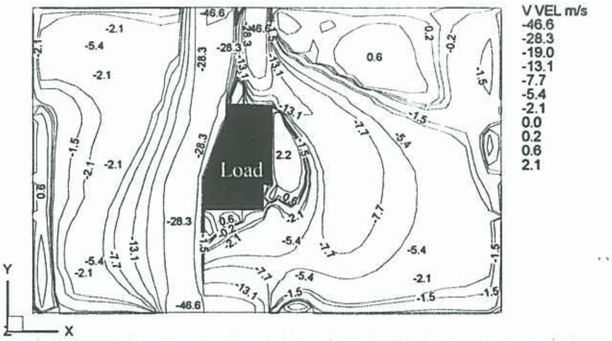


Figure 3(a) Gas velocity contours predicted inside the hot zone in the vertical configuration(Planar cross-section X-Y)

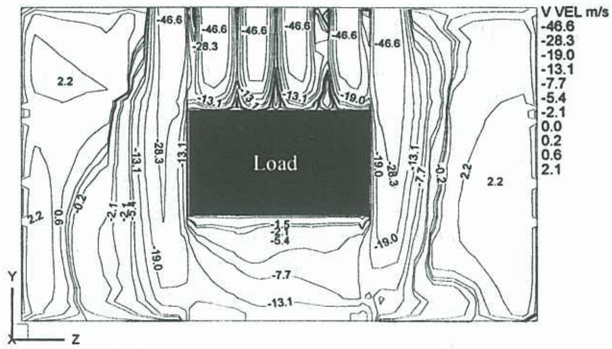


Figure 3(b) Gas velocity contours predicted inside the hot zone in the vertical configuration(Planar cross-section Y-Z)

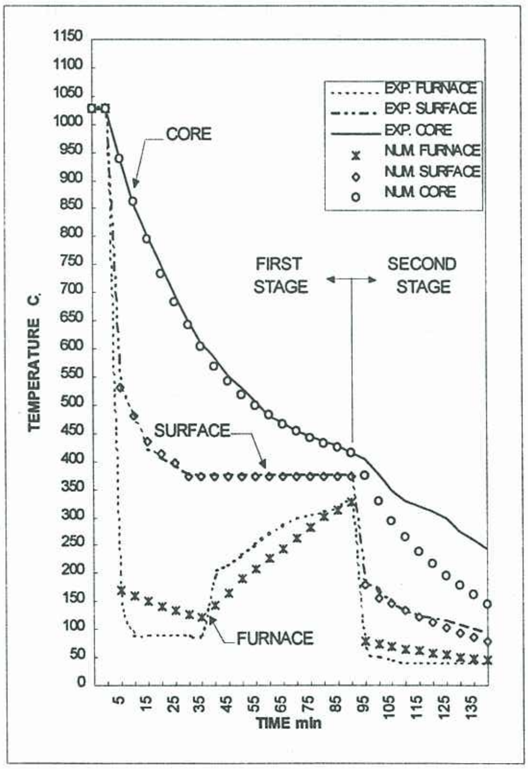


Figure 4 Comparison between the experimental data and numerical prediction

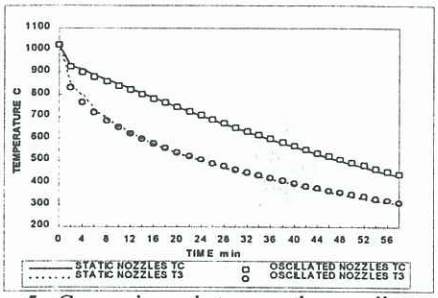


Figure 5 Comparison between the cooling curves predicted by simulation the treated load in the vertical configuration under static and oscillating gas nozzles for two monitored points TC and T3.

CONCLUSION

The computational fluid dynamics approach to fluid flow simulation can give important insights to the gas flow inside the hot zone. It is evident from simulation that there is some loss of gas quench efficiency due to the development of gas recirculation areas within the hot zone which, in turn, influences the temperature profiles generated in the load. The configuration of the load with respect to the gas nozzles in quenching can significantly alter the thermal gradients within the load. With respect to the large die block modelled in this study, oscillating the gas nozzles and alternating the gas quenching from the top and bottom of the hot zone was found to have surprisingly little influence on thermal gradients within the load. Finally, a reasonably close agreement was obtained for cooling curves determined experimentally for marquenching with those predicted by simulation.

REFERENCES

AROLA,R. MARTIKAINEN,H. and VIRTA, J. (1992), Computer Aided Simulation of the Temperature and Distortion of Components during Heating up in Heat Treatment Furnaces, Materials Science Forum, Vols. 102-104, 783-790.

CAVALLARO,G.P., WILKS,T.P., STRAFFORD,K.N., and COOPER, A.R., Heat Treatment Industry Developments, DIST-Regional Development, Canberra

CFX-F3D (1995) Computational Fluid Dynamic, User Manual, CFDS Department, Harwell, UK.

CHEN, X. , ZHANG, J. BLICBLAU, A.S. and DOYLE E.D. (1994), Computer Modelling and Simulation of Gas Quenching in a Vacuum Furnace, Proceedings 1st Biennial Conference on Mathematics in Engineering, Melbourne, July 11-13

RODIC,J. and RODIC, A.(1983),. Computer Aided Heat Treatment and Predicting of Properties, Industrial Heating October , 36-38.

SHAPIRO, A.B. (1984), TOPAZ-A Finite Element Heat Conduction code for Analyzing 2-D Solids, University of California, Lawrence Livermore National Laboratory, USA.

KATATNY, I, JACQUES, P., BLICBLAU, A.S., DOYLE E.D. and MORSI, Y.(1997), Performance and Computer Simulation of Large H13 Dies in Vacuum Heat Treatment, 17th Heat Treating Society Conference, ASM International, Indianapolis, Sept.