Ninth International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
10-12 December 2012

MULTI-SCALE MODELLING OF HIGH PRESSURE GAS FAN QUENCHING FOR GAS TURBINE APPLICATIONS

Francesco COSENTINO*, **Jean-Christophe GEBELIN, Nils WARNKEN and Roger C. REED**

[1] Department of Metallurgy and Materials, University of Birmingham,
Edgbaston, Birmingham, B15 2TT, UK
*Corresponding author, E-mail address: fxc921@bham.ac.uk

ABSTRACT

A methodology for the simulation of heat treatment is presented, involving a multi-scale decomposition approach. It is applied to the high pressure gas quenching operation. The large domain typical of an industrial type furnace is subdivided into smaller domains, each addressing a specific length scale. Considerable reduction in computational time is thus achieved, making it possible to analyse large systems which would otherwise be intractable. The different scales constitute CFD models that are run independently, but are coupled in a hierarchical way *via* suitable choice of boundary conditions. The proposed model is applied to the simulation of gas quenching of turbine blade components. Prediction of cooling rates and temperature distribution is made. Results are compared to a standard full-scale CFD analysis. It is shown that good accuracy can be maintained while shortening the simulation time considerably.

Keywords: Multi scale modelling; Gas fan quenching;

NOMENCLATURE

p	pressure
v	velocity
ρ	density
μ	dynamic viscosity
α	permeability
C	internal resistance factor

INTRODUCTION

Gas fan quenching is a method used for the heat treatment of nickel based superalloys, of the type employed for the turbine blades in the gas turbines used for jet propulsion. It is clean, environmental friendly and can be integrated into vacuum furnaces better than quenching into liquids such as oil. However, it needs particular attention to detail because inhomogeneous cooling rates can be easily produced. For that reason, several examples can be found in the literature where CFD modelling has been used to simulate heat transfer and fluid flow occurring during high pressure gas quenching. For example, analysis of the effect of pressure, gas velocity, impact angle (Elkatatny, Morsi, Blicblau, Das, & Doyle, 2003), furnace load (Macchion, 2004) and for the prediction of heat transfer coefficients (Berglund, Alberg, & Runnemalm, 2003) has been presented.

One characteristic that is common to this research is that, due to the computational cost of large CFD calculations, either simplified geometries of a small number of components were treated. This can often limit the usefulness of the models, because the situation encountered in industry can often involve several hundred components being heat treated at the same time. In practice, large furnaces are used and their characteristic gas flow can be different from those of smaller laboratory-scale equipment. Thus, research is needed to develop techniques that can reduce run times, so that large scale systems can be analysed of the type encountered in practice.

For such large systems, physical processes like fluid flow and heat transfer are usually occurring at very different length scales. Therefore, one possibility is to subdivide the domain according to the length scale of the process in order to treat each scale individually. At each level of decomposition simplifying assumption can be made to reduce the computational effort. Another possibility often encountered in the literature is simplification of the geometry by replacing complex shaped parts with similar bodies of a simpler shape. This approach is particularly suitable when the problem presents a high number of components of the same, complex shape and it has been applied to simulation of printed wired board (Tang & Joshi, 2005) and other electronic equipment (Shankaran, 2002). In the approach used by these authors the large scale simulation with simplified objects is used to generate proper boundary condition for a lower scale simulation with enhanced geometrical features. Another approach, used commonly for heat sinks, is to replace the component by a plain block of similar dimensions, and to assign it effective flow and thermal properties in order to model the effect of the component on the thermal and flow fields (Linton & Agonafer, 1995). In such cases it is necessary to determine the properties of such "porous block"; a technique for doing so has been proposed by Narasimhan both for thermal properties (Narasimhan, Bar-Cohen, & Nair, 2003a) and flow properties (Narasimhan, Bar-Cohen, & Nair, 2003b), either by comparison with full CFD models, empirical correlations or using experimental results.

In this work, a combination of the two approaches is employed to address the problem of gas fan quenching. A three level decomposition approach is utilised similar to (Schmidt & Fritsching, 2006). The flow is solved on the three scales taking into account the effect of turbulence, while the heat transfer problem is only considered in the third, detailed level. Thus the computational time needed is reduced since the main interest is in the prediction of cooling rate at the component level more than the temperature distribution in the furnace. The different scales are coupled by transferring exact boundary

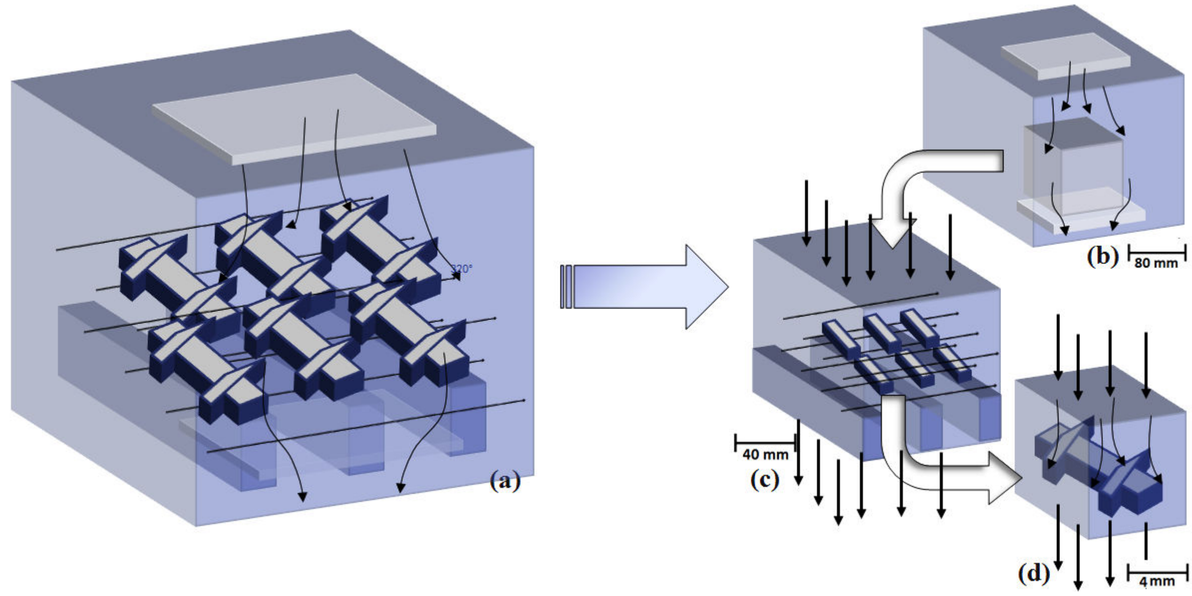

Figure 1: Schematic illustration of the multi level decomposition of the laboratory scale vacuum furnace used in this study.

conditions, considering the contribution of all the six boundaries.

The system analysed is a laboratory scale vacuum furnace where the solution of the full problem is still feasible, so that it can be used to validate the efficacy of the multi scale approach.

MODEL DESCRIPTION

In order to reduce the computational cost associated with a calculation on a very scale, the problem is addressed by subdividing the domain in three levels, each of which is aimed at resolving the flow at a given length scale.

Three separate models are build and run individually, but are coupled by transferring boundary condition in a hierarchical way, from the larger scale to the smaller scale. The flow equations in each of the levels were solved using the CFD software package ANSYS Fluent v13. All the simulations were run taking into account the effect of turbulence by means of the $k - \omega$ model, which explicitly resolves the boundary layer without using wall functions. At each length scale, the flow field is solved in a steady state manner and the transient calculations are performed only for the cooling simulations, where the flow is assumed not to vary.

The scope of the large scale simulation is to resolve the salient features of the flow in order to produce accurate boundary conditions to be fed into the lower scale simulation, and the porous medium takes into account the pressure drop due to the furnace load. At each level, some geometrical details of the full problem are removed to speed up the calculation, but their effect is taken into account to ensure a consistent treatment.

Thus the proposed decomposition consists of three levels (Fig. 1):

i. A *macro scale* model, where the furnace is modelled explicitly but the load is replaced by a plain block of porous medium (Fig. 1-b).
ii. A *meso scale* model, focused on the stack of parts that are modelled using dummy shapes instead of turbine blades (Fig. 1-c).
iii. A *micro scale* model, where only one blade and its surroundings are modelled in full detail (Fig. 1-d).

The first two levels, *macro* and *meso*, are used only to calculate the flow field. This implies that the thermal properties of the porous block, like the effective thermal conductivity, are not needed. The heat transfer calculation is performed directly on the last *micro scale* model to predict the distribution of surface heat transfer coefficient and the cooling rate of the components as a function of the position.

The porous medium

The theory of flow through porous media is used in the first level of decomposition to model the momentum loss occurring to the gas flow in the furnace because of the presence of the specimen and the fixtures (rack, support, etc.).

In the furnace chamber all the components are replaced by a simple plain box, which greatly reduces the geometrical complexity and therefore allows the entire domain to be meshed with a considerable cell count saving. Simulations at this level are aimed at obtaining the overall flow field in the furnace and to establish the flow conditions (in terms of velocity vectors and pressure profiles) at the boundaries of the furnace load.

The pressure drop in the y direction due to flow through the porous medium is described by the Darcy-Forchheimer equation (Bejan, 1995):

$$\frac{\partial p}{\partial y} = \frac{1}{\alpha}\mu v_y + \frac{1}{2}C\rho|v|v_y \approx \frac{1}{\alpha}\mu v_y + \frac{1}{2}C\rho v_y^2 \quad (1)$$

This relationship expresses the fact that the pressure drop is a function of two contributions: one is a viscous loss, which is directly proportional to the velocity and the other is an inertial loss proportional to the velocity squared. Two parameters determine the behaviour of the porous medium and those are known as permeability, α, and internal resistance factor C. Since the medium is in general anisotropic, values of α and C for each direction are needed; however, for this application, the flow is almost entirely parallel to the *y* direction (top to bottom flow) so that contributions from the velocity component in the *x* and *z* directions can be neglected, In this case the second term becomes proportional to v_y^2.

To find an appropriate value for these, the meso scale model is used; several simulations are run with a known pressure drop applied, and the resulting mean velocity inside the domain is calculated; the characteristic $\partial p/\partial x$ vs. v is then plotted and, by fitting Eqn. 1 to these points, the values of the constants are then determined. The simulated values used for the fitting are reported in Table 1 (first three columns). The values obtained are $\alpha = 2.4 \times 10^{-3}$ m^2 and $C = 6.43$ m^{-1}.

To double check the validity of this approach, a similar procedure was repeated but this time simulations were run with a uniform velocity profile at the inlet, and the resulting pressure drop was calculated from the simulation results. Computing the curve fitting on this set of data (Table 1, last three columns) yields the following value for the constants: $\alpha = 1.88 \times 10^{-4}$ m^2 and C = 6.5 m^{-2}.

First Approach			*Second approach*		
Pressure drop [Pa]	*Mean v [m/s]*	*Mean v_y [m/s]*	*Pressure drop [Pa]*	*Mean v [m/s]*	*Mean v_y [m/s]*
10	2.19	2.14	10.6	2.24	2.18
20	3.1	3.03	16.47	2.79	2.72
30	3.79	3.71	23.76	3.35	3.27
40	4.39	4.28	32.39	3.91	3.81
50	4.9	4.79	42.26	4.47	4.36
60	5.37	5.24	53.6	5.03	4.9
70	5.8	5.66	66.18	5.59	5.45
80	6.2	6.06	79.94	6.15	5.99
90	6.58	6.42	95.20	6.71	6.54
100	6.93	6.77	111.55	7.27	7.08

Table 1: Pressure drop and corresponding velocities used to characterise the porous medium.

Although the prediction of *C* agrees well with the previous estimate, the prediction of *α* differs by one order of magnitude. In practice it was found that the viscous term has little effect on the characteristic of the flow and in fact the first term in Equation 1 can be completely neglected. This is illustrated in Figure 2 where the modelling results are plotted together with the fitting using *(a)* the Darcy-Forchheimer law, *(b)* the viscous term only and *(c)* the inertial term only. The simulated results show that there is a quadratic relationship between pressure drop and mean fluid velocity; fitting these data using the inertial term only yields the same result as the Darcy-Forchheimer equation. This proves that the viscous term is not relevant and can be neglected.

Model set-up and boundary condition coupling

For the macro scale simulation the pressure drop between inlet and outlet was measured using a differential pressure transmitter. These values were specified uniformly at the inlet and outlet. The porous block is characterized by the constants *α* and *C* and by the porosity which is defined as the volume fraction of fluid within the porous region.

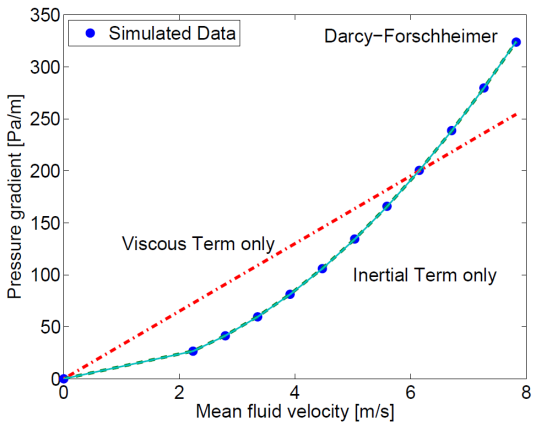

Figure 2: Pressure Drop v Mean velocity. Blue dots represent the simulated data (same as Table 1). The curves represent the three fitting option: Darcy law, Viscous or Inertial term only.

The boundary of the porous block does not need particular attention. Its surfaces are defined as *interior*, a boundary condition that defines surfaces that do not interact with the gas flow, but it allows the recording of and transfer flow quantities to the micro scale model. Once this information has been supplied, the flow field inside the furnace can be calculated taking into account the pressure drop effect of the furnace load (Figure 3). At the six boundaries of the porous block, flow quantities (profiles of pressure and velocity) are exported.

In the meso scale model those quantities are used to define the boundary conditions. The choice of the type of boundary condition to use should not change the simulation result; however it will influence the predicted flow field and therefore an assessment needs to be made of the impact of this coupling. In this work, the following choice was made:

i. On the top surface of the box, always considered as an inlet, velocity components are transferred.
ii. The bottom surface is always considered as an outlet and the static pressure is transferred using a pressure-outlet condition.
iii. For the side surfaces, coupling has been performed with velocity vector components.

Since the flow in the furnace is generally turbulent, also the turbulence quantities, turbulent kinetic energy *k* and specific dissipation rate ω, have been mapped and interpolated from one scale to the other.

Once the meso scale simulation is complete, the flow through the components, modelled as simple shaped blades (Figure 4) is known. A box of 6 *interior* surfaces bounds each blade.

In a similar way as done in the previous step, the flow quantities from one model are assigned to the micro model and the detailed flow field around a component with real shape are thus obtained. Once the flow calculation has converged, the energy equation is activated and the transient heat transfer problem can be solved yielding a prediction of the cooling rate of the blade (Figure 5).

It is important to note at this point that blades positioned in different locations in the load can be simulated by using different boundary condition. All the boundary conditions come from the meso scale model, but it is not necessary to analyse in detail each of the blades in the load. The detailed simulation on the micro scale level will be conducted on selected blades, for which the cooling rate or the temperature distribution is of particular interest.

RESULTS

Multi scale modelling

Figure 3 shows the velocity magnitude field in the macro level simulation. It can be seen how the flow from the top channel changes direction as it enters the furnace chamber. In that region the cross section is smaller and an increase in velocity magnitude is observed. Also it can be noted how the baffles direct the flow towards the back of the furnace, leaving the front part with a very weak flow.

Figure 4 shows the velocity magnitude on the meso scale model. The boundary condition of this calculation comes from the macro model and in fact it can be seen that the characteristic angle at which the flow enters in the domain is retained between the models. At this scale we can also see some recirculation that takes place in the wake of the dummy blades.

In Figure 5 the result of the micro scale model is shown. In this model the detailed flow around a real geometry component is calculated as well as the temperature distribution during quenching. The temperature here shows that during quenching the aerofoil, which is the thinnest section, is quenched more rapidly than the root, which is by comparison very thick. The cooling rate at three selected locations is plotted in Figure 6 and it shows how different the cooling rate can be from point to point in the component.

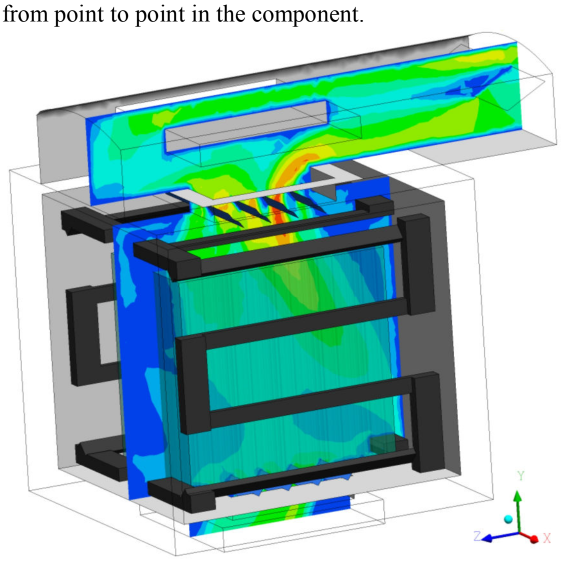

Figure 3: Velocity magnitude field on an x-cut of the macro model of the furnace. The fixtures like rack and support, as well as the specimen are missing, replaced by a porous medium.

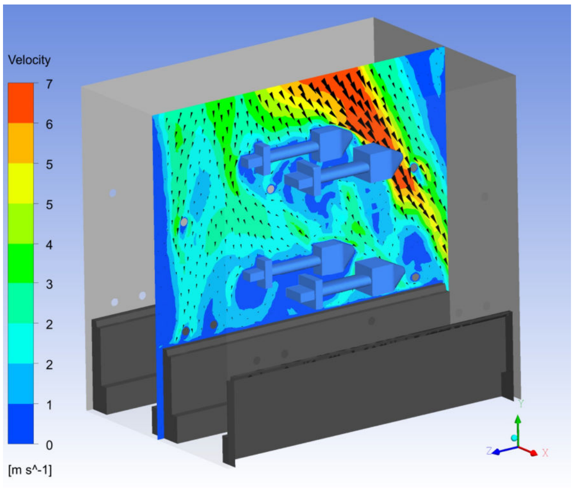

Figure 4: X-cut plane that shows velocity magnitude field within the furnace load. In the meso scale model the load is made up of blades with a simplified shape.

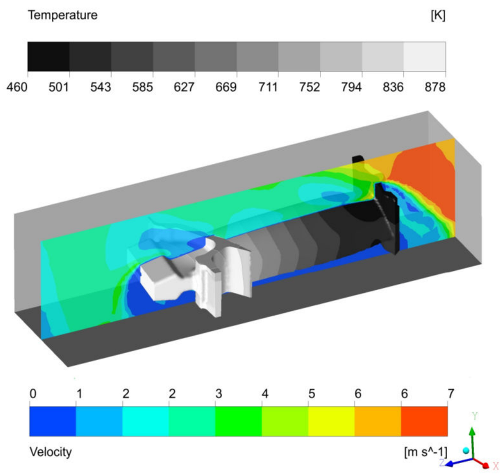

Figure 5: Detailed flow field around a turbine blade, modelled individually (micro scale model) and associated temperature distribution on the surface.

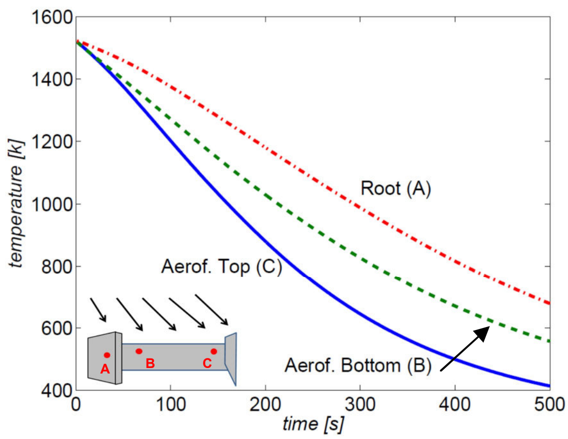

Figure 6: Cooling curve prediction for different locations inside the turbine blade.

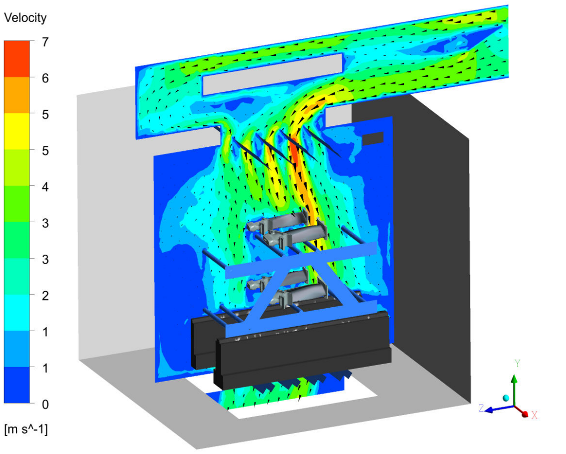

Figure 7: Velocity magnitude in a furnace loaded with 6 blades. Full scale model where every component is explicitly modelled.

Comparison with full scale model

The prediction of the cooling rate made using the multi scale model has been compared with a simulation employing the traditional approach in which the entire model is built and meshed starting from CAD files. The resulting mesh, for the model including only six blades, was almost 10 million elements, even if localised refinement was used on the surface of the blades and near the baffles.

Figure 7 shows the velocity profile on an x-cut in the furnace. Comparison with Figures 3 to 5 shows that the predicted velocity magnitude field is in good agreement with the meso and micro scale prediction, both qualitatively and quantitatively.

In Figure 8 the cooling rates in different blades and at different locations are plotted for the two models. The first graph, Figure 8-a, shows the cooling curve of a blade which is located in the top row, in the centre of the furnace. Because it is directly exposed to the flow field, with little or no shading at all, it experiences a higher cooling rate compared with another blade that is located on the second row and on a side (Figure 8-b).

In both cases, the aerofoil is quenched more rapidly, because of its smaller thickness and the difference is roughly the same.

The agreement between the two modelling approaches is not perfect, but it is satisfactory given the simplification involved and the significant savings in computational time as will be presented in the following section.

Advantages of the multi scale approach

The obvious advantage in the application of this approach is the reduction in calculation time. In the case of the large system of interest here which contains hundreds of blades, analysis would not be possible otherwise.

All the calculations in this work were run in parallel on 8 processing cores; while the full scale model took 55h to complete, the multi-scale model took a total of only 8h.

The introduction of simplified geometries makes the whole CFD process smoother; it is easier to generate high quality meshes and the result files for post processing are smaller in size and easier to handle.

Finally, the boundary conditions applied to the meso and micro scale simulations are well posed, since they come from a previous solution; this eases convergence which is then generally faster.

CONCLUSIONS

A methodology for the simulation of the large scale gas fan quenching has been presented. It has been applied to the simulation of quenching of turbine blades, of the type needed for jet propulsion applications. The reduction technique is based upon the decomposition of the domain in three levels. The main conclusions that can be drawn from this work are:

- A combination of two approaches – porous medium and geometry simplification – has been demonstrated; it involves the decomposition of the domain on multiple length scales.
- The application of porous media theory is effective in replicating the pressure drop due to the interaction of the fixtures and components with the flow field.

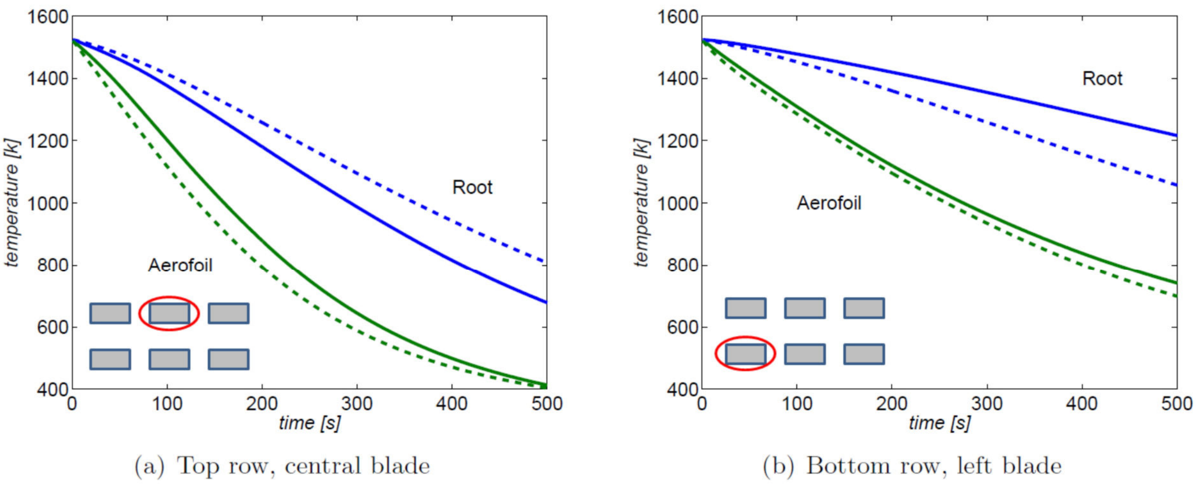

Figure 8: Quenching prediction of the full scale simulation (dashed line) compared with the multi scale simulation (solid line). Each graph represents a blade in a different position in the furnace (sketched in the bottom left corner). In each case, two locations (root and aerofoil) have been investigated.

- Porous medium can be characterised using the meso scale model, which contains all the bodies located in the furnace, by fitting the Darcy equation.
- The predicted cooling curves from the multi-scale model compares well with the full scale simulation.
- The reduction in time needed to perform the calculation is significant; this enables the simulation of a large scale system that could not be treated otherwise.
- Detailed simulation is performed only on selected blades within the load, making the analysis more efficient.

ACKNOWLEDGEMENTS

The authors are grateful to the Engineering and Physical Sciences Research Council (EPSRC) of the United Kingdom for funding under the SAMULET project. Rolls-Royce plc is acknowledged for the provision of the materials used for this study.

REFERENCES

BEJAN, A., (1995), “Convective heat transfer (2nd ed.)”, *Chichester: Wiley.*

BERGLUND D., ALBERG H., and RUNNEMALM, H. (2003), "Simulation of welding and stress relief heat treatment of an aero engine component". *Finite Elem. Anal. Des.*, **39(9)**, 865–881.

ELKATATNY I., MORSI Y., BLICBLAU A. S., DAS S., and DOYLE E. (2003), “Numerical analysis and experimental validation of high pressure gas quenching”, *Int. J. Therm. Sci.*, **42(4)**, 417–423.

LINTON R. L., and AGONAFER D. (1995), “Coarse and detailed CFD modeling of a finned heat sink”, *IEEE Trans. Compon. Packag Manuf. Technol. Part A*, **18(3)**, 517-520.

MACCHION O. (2004), “Computational study of velocity distribution and pressure drop for designing some gas quench chamber and furnace ducts”, *J. Mater. Process. Technol.*, **155-156**, 1727-1733.

NARASIMHAN S., BAR-COHEN A., AND NAIR R. (2003), “Thermal compact modeling of parallel plate heat sinks”, *IEEE Trans. Compon. Packag. Technol.,* **26(1)**, 136-146.

NARASIMHAN S., BAR-COHEN A., AND NAIR R. (2003), “Flow and Pressure Field Characteristics in the Porous Block Compact Modeling of Parallel Plate Heat Sinks”, *IEEE Trans. Compon. Packag. Technol.,* **26(1)**, 147-157.

SCHMIDT R. R., and FRITSCHING U. (2006), “Homogenization of hardness distribution and distortion in high pressure gas quenching”. *Materialwiss. Werkstofftech.*, **37(1)**, 103–109.

SHANKARAN G. (2002), “A multi level modeling approach with boundary condition import for system, sub-system or board level thermal characterization”, *Thermal Measurement and Management*.

TANG L., and JOSHI Y. (2005), “A Multi-Grid Based Multi-Scale Thermal analysis approach for combined mixed Convection, Conduction, and Radiation due to discrete heating”, *J. Heat Transfer*, **127(1)**, 18.

