M. Venturino, P.A. Rubini. "Coupled fluid flow and heat transfer analysis of steel
reheat furnaces". 3rd European Conference on Industrial Furnaces and Boilers.
Lisbon, Portugal, 18-21 April 1995. Eds. Leuckel, Collin, Ward, Reis. INFUB,1995,
ISBN 972-8034-02-4

Coupled Fluid Flow and Heat Transfer Analysis of Steel Reheat Furnaces

M. Venturino and P. Rubini
School of Mechanical Engineering
Cranfield University, Cranfield, England

Abstract

A computational methodology to predict the transient heat transfer to the load in a continuous steel reheating furnace is presented. The procedure couples a numerical solution of the fluid flow and the thermal radiation inside the furnace, obtained using conventional Computational Fluid Dynamics techniques, to a numerical solution for the temperature inside the steel charge.

The conduction module solves for the transient heat transfer to the steel slabs, allowing for movement inside the furnace during the reheating operation, using a fine grid embedded in the fluid domain. The full transient problem is simplified by assuming a steady state furnace atmosphere through which a continuous series of identical stock are moving. This procedure allows for an economic solution for the flow field and thermal radiation without losing details of the conduction within the slab.

The resulting model has been applied to a three dimensional steel I-Beam moving through a furnace with a predefined gas temperature, uncoupled from the CFD prediction of the flow field. The fully coupled procedure has then been used to study a steel reheating operation for a two dimensional model furnace with rectangular slabs.

Keywords

Mathematical modelling; Computational Fluid Dynamics; Coupled heat transfer: Reheating furnaces; Walking beam furnaces

Introduction

Steel reheating represents a critical step within the steelmaking mill process. Steel reheating furnaces are used to heat slabs and billets to a suitable temperature for the following rolling operation. The amount of energy required to raise the temperature of the load a few hundred degrees is considerable, therefore economic pressures have directed a significant effort towards maximising furnace capacity whilst minimising energy usage.

The goal of a reheating furnace is to heat the steel charge to the minimum temperature consistent with achieving the correct temperature and metallurgical properties at the finishing stands of the mill. Uniformity of the temperature within the load, minimisation of local temperature gradients, avoidance of surface defects such as skid marks, overheating marks and oxidation scale represent the characteristics of the ideal product of the reheating operation.

In the last two decades the principal lines of research in the steel reheating industry have been developments of innovative furnaces, such as inductive and resistance furnaces and the improvement of traditional designs together with the massive introduction of automatic control. In all of these processes a primary role has been played by mathematical modelling which represents, together with physical modelling, a fundamental tool to assist every development in this field. Due to the complexities of the physics that occurs inside traditional steel reheating furnaces many mathematical models have been proposed over the past years. The complexity of the models has roughly paralleled the then available computing power.

Early models of steel reheating furnaces only solved for heat conduction within the load by employing simplifying assumptions about the gas temperature within the furnace and heat transfer coefficients derived from known furnaces, [1,2]. These methods have been used for evaluating the effect of modifications on existing furnaces.

A number of models have been developed which make use of the zone method for the solution of the thermal radiative heat transfer inside furnaces, [3,4,5]. Computational model based on the "zone assumption" vary in complexity depending on the number and arrangement of zones that subdivide the furnace (simple gas zone, long-furnace model, multidimensional zone model). In all of these models the effect of the furnace atmosphere, for example flow field and combustion have been almost neglected. Whenever required, data concerning the flow field and combustion have been provided from other sources, typically experimental results or less commonly results from Computational Fluid Dynamics calculations.

More recent studies have investigated the flow and heat transfer within furnaces by solving conservation equations using Computational Fluid Dynamics techniques, [6,7,8]. Due to the complex physics inside the furnace such as turbulence, combustion and thermal radiation, a different set of approximations must be made. Typical studies have been concerned with the solution of only the isothermal flow field showing, therefore, the effect of the gas path on the performance of individual furnaces. Computer memory demand, large run times and difficulty of use have restricted the application of traditional CFD in full furnace modelling. Moreover the wide range of characteristic dimensions in a furnace renders a conventional CFD approach almost impossible to resolve the fine details of the conduction process simultaneously with the flow field.

Method

The heating operation for an individual slab usually takes between two and three hours during which time the slab is moved inside the furnace in a prescribed manner dependent upon the type of furnace and operation procedure, for example walking beam or pusher furnace.

Within the furnace the slab experiences different heat transfer conditions dependent upon the surrounding atmosphere. The external conditions directly modify the heat flux to the stock via the gas temperature and the convective heat transfer coefficients, hence they are fundamental in determining steel properties and phenomena such as scale formation.

The physical problem of a continuous reheat furnace has been approached from the generic viewpoint of coupled heat transfer between a fluid phase and a solid phase. The fluid phase is modelled using the commerical CFD package, FLUENT [9], and is coupled to a specifically written transient conduction solver. The CFD code solves the modelled three dimensional Navier-Stokes equations over a boundary fitted grid using a finite volume pressure correction procedure. For turbulent reacting flows the k-ε turbulence model is employed in conjunction with an eddy-break combustion model, thermal radiation is modelled using the Discrete Transfer Radiation Model, a ray tracing methodology. The present work is solely concerned with the development of the coupled, embedded grid, conduction solver, for further details of the CFD techniques employed see for example [10].

For the solid phase (steel charge) a control volume technique is employed to discretise and solve the differential heat conduction equation, which may be expressed in general non-orthogonal curvilinear coordinates as,

$$\rho c \frac{\partial}{\partial t}(T) = \frac{\partial}{\partial \xi_j}\left(J\, k\, g^{jk} \frac{\partial T}{\partial \xi_k} \right) + J\, S(\xi, \eta, \zeta)$$

where g^{ij} represent the contravariant metric tensor components and J is the jacobian determinant for the coordinate transformation. Spatial and temporal discretisation have been achieved using a second order and a fully implicit first order accurate scheme respectively. The solution procedure to solve this system is based on a combination of a line by line tri-diagonal matrix algorithm (TDMA) and a block correction scheme. General Dirichlet and non-Dirichlet boundary conditions are allowed for. The solutions for the fluid flow and the solid temperature are coupled at their common boundaries by the incident radiative heat flux and similarly by convective heat transfer.

A full simulation of an operational furnace would have to allow for both the movement of the load within the furnace and for the transient nature of the furnace atmosphere, both in terms of the fluid dynamics and the heat transfer. A full simulation is possible, but since typical transient time steps for the fluid phase would be of the order of seconds, for at least token accuaracy, and the residence time for a single slab is of the order of three hours, the computation would be very expensive and time consuming to obtain.

The approach adopted in the present work is to assume that the furnace is operating in a pseudo steady state, where the fluid flow is at steady state and a continuous stream of slabs or billets pass through the furnace. When a single slab is the point of reference then a transient simulation is necessary to simulate the heat transfer process. This simplification is deemed justifiable by the considerable thermal inertia of a furnace, such that for suitably long periods of continuous operation, minor fluctuations in the fluid phase due to burner control and the movement of the load itself may be ignored.

For a walking beam furnace the slab reheating operation consists of a repeated sequence of movement and then delay in a prescribed position and of prescribed duration. In the present model movement from one position to the next has been considered instantaneous. The reheat operation for the charge is then modelled as one of transient heat conduction with important step variations at the boundaries occurring at each slab move. During the flow field solution the furnace load is represented by fixed blockage regions with Dirichlet temperature boundary conditions.

The procedure to obtain the overall solution can be represented as an alternate sequence of calls to the flow field solver and the conduction solver, see figure 1. Within the call to the conduction solver a single slab is followed during its sequence of movements and delays inside the furnace from the initial position to the exit position. The transient heat transfer to the slab during the reheating operation is calculated considering a fixed heat flux boundary condition obtained from the previous CFD flow field calculation. On termination of the slab calculation a new loop of fluid flow calculations are performed considering for every blockage region a fixed boundary temperature obtained from the last predicted skin temperature of the slabs.

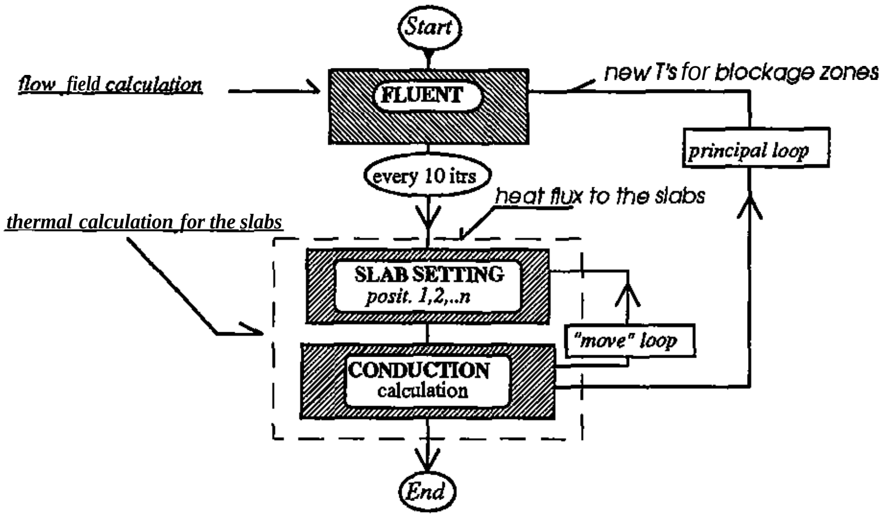

Figure 1: Coupled Flow and Slab Calculation Scheme

A particularly novel feature of this work is that the program allows computational meshes of different sizes to be used for the fluid and solid regions. The procedure may be described as an embedded grid solution with sliding block meshes. The furnace space and the load are discretized using separate grids at which nodes the solutions of the respective equations are obtained. The two meshes are created independently of each other and are only related by the region of the furnace on which they overlap and the degree of accuracy required for the solution. Typically the mesh for the furnace is much coarser than that of the solid, consequently interpolation and extrapolation is required to transfer information between the two grids.

The solution for the load temperature is obtained utilising boundary conditions derived from the flow field solution, convective heat flux from the fluid flow calculation and radiative heat flux from the radiation model. These data are not directly used as fixed superimposed boundary conditions. Consider the convective heat flux, whilst the slab is warming up the changing surface temperature produces a consequent change in the net heat flux from the surrounding furnace atmosphere. A fixed value of heat flux would be unrealistic and could cause convergence problems.

This problem, which is a result of assuming a steady state flow field, has been circumvented by utilising the fluxes to determine local instantaneous correlated properties of the heat transfer process. From the convective heat flux data an estimation of the convective heat transfer coefficient is worked out. The heat transfer coefficient and the temperature of the flow are then used as fixed values during the conduction calculation to compute the real heat flux to the boundary cells of the slab. Analogously the radiative heat flux allows an apparent external radiative temperature to be estimated for each cell of the slab surface. The apparent radiative temperature and convective heat transfer coefficients are determined prior to the transient conduction solution.

At the end of the conduction calculation, when the slab has been marched through the length of the furnace, the surface temperature of the slab at each position within the furnace is used to update, by interpolation, the prescribed temperature of the blockages within the fluid flow calculation. The following CFD iterations, typically ten or twenty before the next call to the conduction routines, assume the blockage temperatures to be Dirichlet values. The fluid flow solver calculates new convective and radiative fluxes which will be used as the new boundary conditions for the next call to the conduction routines. The whole process is repeated iteratively until the solution converges. Convergence is defined within the flow solver in terms of the absolute mass error within the solution and in the solid as the net change in temperature.

Applications and Results

Validation of the Conduction Solver

The conduction solver was first validated against an analytical solution, [11,12]. The principal aim of the comparison was to evaluate the dependency of the error upon the time discretization. The results of this exercise are show in in figures 2 and 3 for the one dimensional problem of a solid with Dirichlet boundary conditions, of $T=0$, imposed at $x=0$ and $x=L$. The solid is initialised with a constant temperature profile, $f(x)=T_0$, and the resultant relaxation to the steady state solution is observed.

Figure 2 presents a comparison of the temperature distribution inside the slab for different instants in time, represented in non-dimensional form by the Fourier number, $F_o = at/L^2$.

Figure 3 presents an analysis of the relative error for different values of the time step (Fourier number). The relative error is defined as, $(T - T_{exact})/(T_{max} - T_{min})$ where T_{max} and T_{min} are the maximum and the minimum temperatures in the temperature field. As expected, it can be observed that the relative error has the maximum value at low Fourier numbers where it is sensitive to the effect of the initial temperature step, after that it decreases quickly.

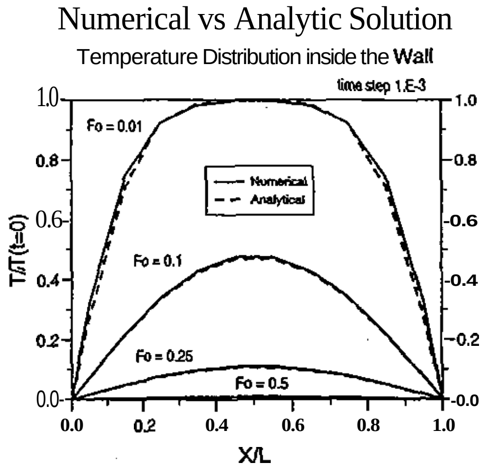

Fig.2: The Test Case. The Thermal Transient

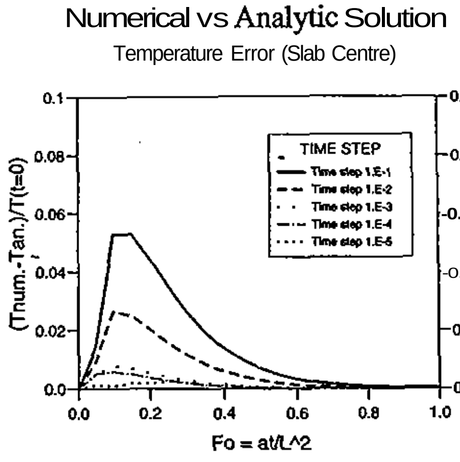

Fig.3: The Test Case. The Error Evaluation

Application to an I-beam

As a first application a 2 hour 45 minute reheating process for an "I" beam has been simulated. The application demonstrates the coupling procedure between the heat conduction routine and slab movement routine [11].

The furnace load was modelled as a steel "I" beam, moving through the furnace in five minute stages. The solution grid for the beam was calculated using a simple algebraic procedure and consists of approximately 7700 node points.

The furnace was modelled as a rectangular section duct, 30m by 12m by 8m, subdivided into three distinct heat transfer zones (8m, 12m, 10m), each with a constant gas temperature of 600K, 1500K and 1200K respectively. In the final zone the actual heat flux to the beam was prescribed to be zero to simulate the effect of a soak zone. Adiabatic boundary conditions were prescribed for the two surfaces of the beam assumed to be in close proximity to each other in the furnace.The external convective heat transfer coefficient was prescribed to be 30 W/m^2k and the surface emissivity to be 0.8 W/m^2K^4.

The surface oxidation of the steel and and the formation of scale is taken in to account by a simple Arrhenius based model. An expression for the rate of growth is considered dependent on an activation temperature, *B*, and a pre-coefficient, *A*, where L is the scale thickness.

$$\frac{dL}{dt} = \frac{K_T}{L}, \quad \text{where } K_T = A\, e^{-\frac{B}{T}}$$

Typical values for the Arrhenius coefficients were 14410 *k* and 7.1×10^{-6} m^2/s for the activation temperature and the pre-coefficient respectively. These values are optimised values obtained by numerical tuning of the parameters. The conduction program was run on a DECStation 5000/200, the typical CPU time was about 260 seconds for the I beam simulation. Full details of the simulation are available in Venturino [11].

Figures 4 and 5 present the steel temperature and net heat flux to the beam respectively as a function of time within the furnace. The temperature profile of the beam is seen to follow a typical heat profile for a reheat furnace. Figures 6 and 7 show the temperature distribution within the steel during the middle of the reheating operation. This calculation is a demonstration only, no analytical or experimental data is available for comparison.

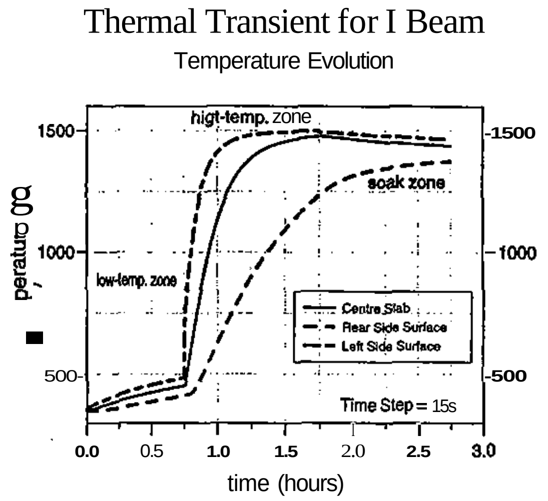

Fig.4: The I-Beam Case: Thermal Transient

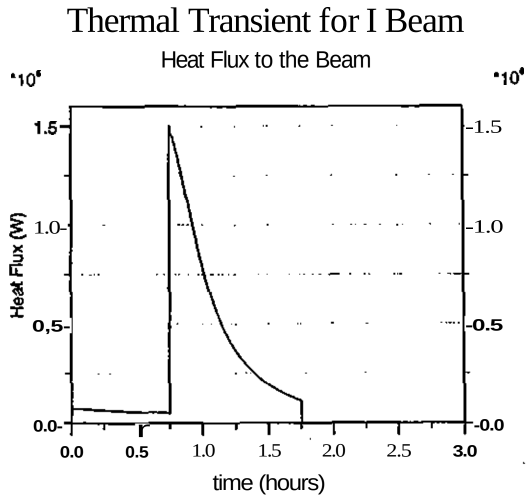

Fig.5: The I-Beam Case: Heat Flux

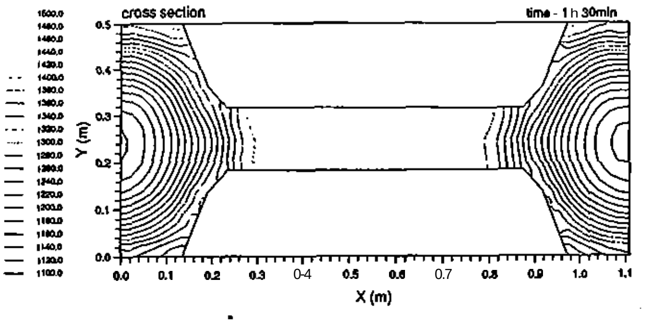

Fig. 6: The I-Beam Case: Temperature at the Cross Section

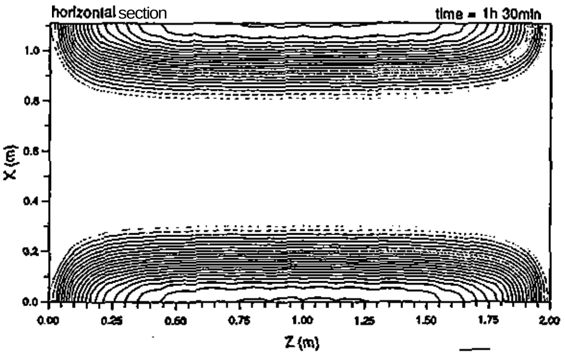

Fig.7: The I-Beam Case: Temperature at the Horizontal Section

Application to a two dimensional furnace

The fully coupled procedure has been applied, as a first demonstration, to the simple application of a two dimensional furnace reheating a rectangular slab. Slabs of dimensions 1.2m by 0.4m enter a 12.5m long furnace zone as shown in figure 8. The slabs are assumed to be at an initial temperature of 350K. The reheating process for each slab lasts one hour and the slabs stop six times along the furnace for a duration of 12 minutes each time. The furnace zone is heated by hot combustion products being exhausted from a single burner at a temperature of 1500K and a velocity of 3 m/s. The hot gases pass through the furnace and exit from an outlet on the floor of the furnace. The walls of the furnace are considered adiabatic and a value of emissivity of 0.8 is prescribed.

The fluid domain was discretised using a cartesian mesh of 4635 nodes and for the slab 11x21 grid points were used. This grid resolution allowed the slab to be modelled with four times as many grid nodes as the corresponding fluid flow grid would have allowed. The radiative heat transfer inside the furnace was modelled using the Discrete Transfer radiation model with 16 rays for each radiating surface.

Figures 9 and 10 illustrate the predicted flow field and furnace gas temperature respectively. Points to note are the low velocity region at the discharge side (left) of the furnace and an almost uniform gas temperature in the lower section of the furnace. Figures 11 and 12 illustrate the steel temperature and the net heat flux to the steel respectively throughout its duration within the furnace. The temperature-time history for the slab again shows the typical profile expected for a reheating furnace, the soak zone at the end of the furnace is this time simulated by the absence of a burner. The transient variation of the heat flux to the slab is in accord with the predicted gas temperatures, peaking in a region just downstream of the burner exit.

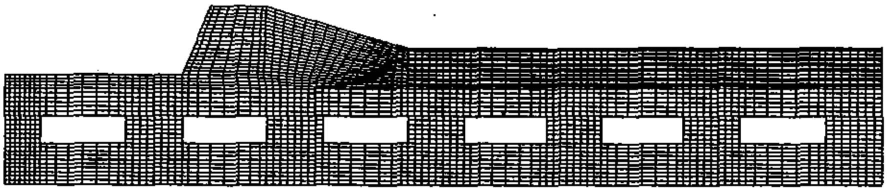

Fig.8: The Furnace Grid and Slab Positions

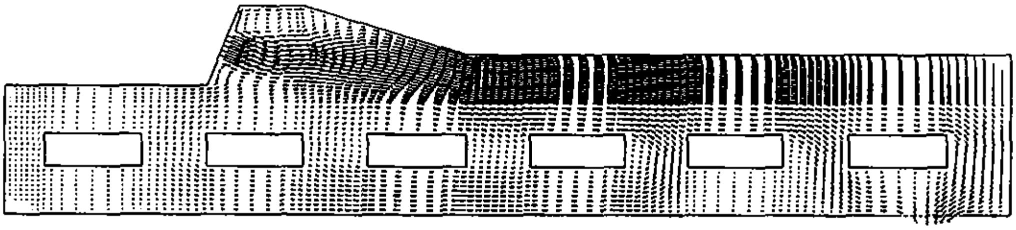

Fig.9: The Flow Field

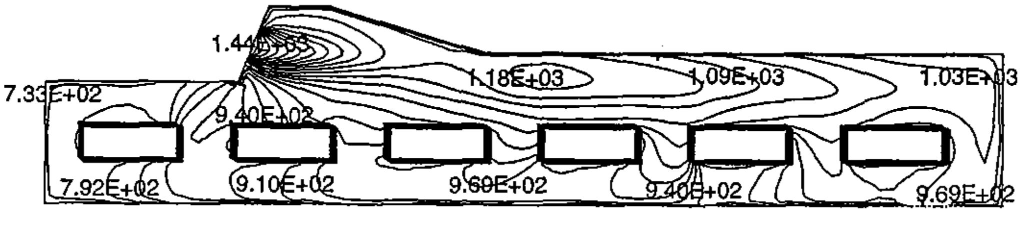

Fig. 10: The Temperature Field

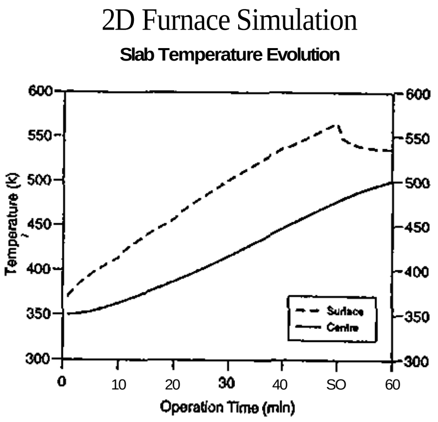

Fig. 11: The Thermal Transient for the Slab

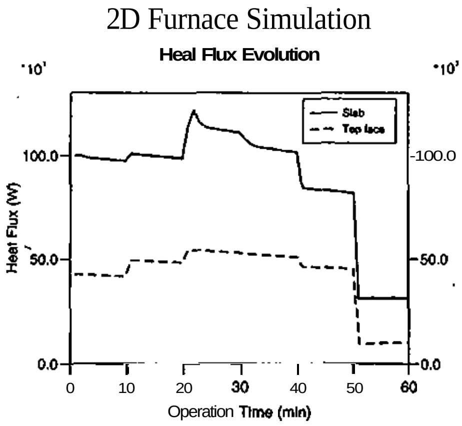

Fig. 12: The Heat Flux Transient for the Slab

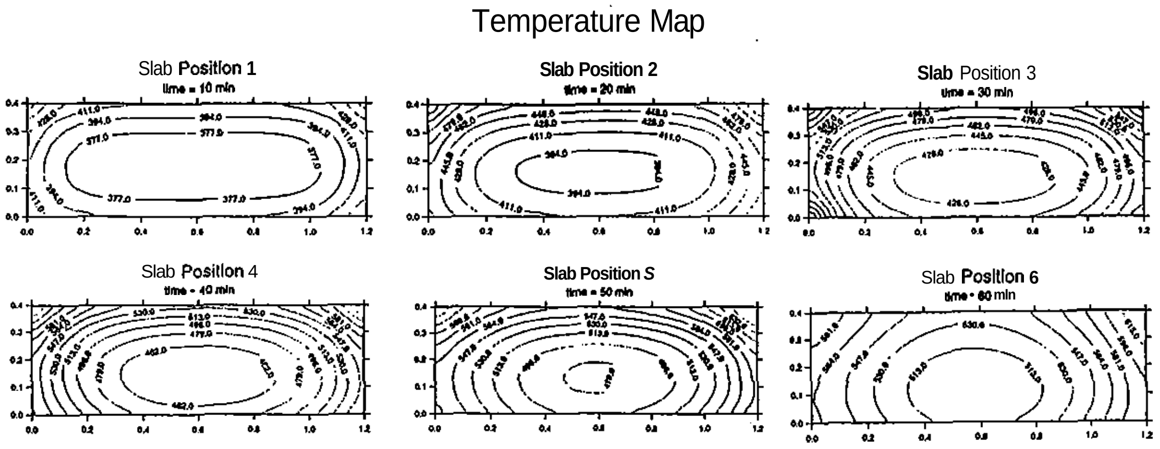

Fig. 13: The Variation of the Slab Temperature

Conclusion

The above examples have demonstrated the accuracy and application of a coupled fluid flow and conduction solver combining embedded grids and sliding blocks. The accuracy of the conduction solver has been demonstrated by comparison with an analytical solution. The utility of the general curvilinear coordinates have been demonstrated for the conduction solver by the application to an I-Beam. The fully coupled procedure has been demonstrated by application to an example furnace zone. One of the principal advantages of the present work is that an application specific module has been coupled to a commercial CFD package, this significantly reduces the development time of the full coupled procedure.

The next stage in the development of the coupled modelling procedure is the extension to three dimensions. Once complete a full validation can commence by comparison with data obtained from operational furnaces and from existing, empirically based, furnace modelling programs. At this stage the true furnace geometry and boundary conditions must be accounted for, including, for example, the charge and discharge doors, skids and skid supports and realistic non-adiabatic furnace walls.

Once validated, the full utility of a coupled CFD and thermal analysis program can be applied to the investigation of physical phenomena of significant practical importance, for example the the minimisation of skid marks, temperature non-uniformities and surface oxidisation.

References

[1] F.Fitzgerald, A.T. Sheridan, *"Heating of a Slab in a Furnace",* J.of Iron and Steel Institute, Vol.208, 1970

[2] R. Collin, *"A Flexible Mathematical Model for the Simulation of Reheating Furnace Performance",* Proc. of Conference for Hot Working, 1968

[3] Hottel H.C., Sarofim A.F. : *" Radiative Transfer "* MacGrow-Hill, N.Y. (1967)

[4] Shih H. T., Ho T.Y.: *"Study of the Total Heat Exchange Factor in the Reheat Furnace by the Zone Method"* Proceedings of International Symposium on Steel Reheat Furnace Technology Ontario- Canada -(1990)

[5] Selcuk N. : *"Evaluation of Flux Models for Radiative Transfer in Rectangular Furnaces "* Int. J. Heat and Mass Transfer, Vol. 31, No 7 -(1988)

[6] Chen J. D., Ho T. Y. : *"Numerical Analysis of Isothermal Flow Fields in Modelled Reheating Furnace"* Proceedings of International Symposium on Steel Reheat Furnace Technology Ontario- Canada -(1990)

[7] Hug P. : *"Improvement of Reheating Homogeneities of Slabs by Control of the Gaseous Flow Patterns "* Proceedings of International Symposium on Steel Reheat Furnace Technology Ontario- Canada -(1990)

[8] P. A. Rubini, H. A. Becker, E. W. Grandmaison, A. Pollard, A. Sobiesiak *"Three dimensional Modelling of a Steel Reheating Furnace",* Proc. Intern. Simposium on Steel Reheat Furnace Technology, Canada, 1992

[9] FLUENT " *User's Guide - Version 4.2* "

[10] Venturino M. *"The Coupling of a CFD Program and a Conduction Solver for Steel Reheating Furnace",* Cranfield University Internal Report, 1994

[11] Venturino M. *"The Steel Reheating Furnace Modelling",* Cranfield University Internal Report, 1993

[12] H. S. Carslaw, J. C. Jeager *" Conduction of Heat in Solid",* Oxford Univ. Press, 1959