11th Australasian Fluid Mechanics Conference
University of Tasmania, Hobart, Australia
14-18 December 1992

5B-3

CALIBRATION OF THE FLUID FLOW ASPECTS OF A CONTINUOUS CASTING MOULD NUMERICAL MODEL

Paul J. FLINT, Qinglin HE, Rama B. MAHAPATRA and Joe HERBERTSON

BHP Research, PO Box 188
Wallsend, NSW 2287, AUSTRALIA

ABSTRACT

A CFD model has been developed to describe the heat transfer, solidification and fluid flow in the steel slab casting mould. The model is three-dimensional and steady-state, allowing determination of the effect of various operating variables on solidification rate, and other product quality issues. This paper concentrates on the model tuning procedure, using data from full-scale water modelling experiments. The model parameters considered include mesh density, inlet velocity profiles, inlet turbulence boundary conditions, and turbulence model parameters. As well, two discretisation schemes are compared, and two types of wall boundary conditions are compared with experiment. The principal conclusion is that the numerical model is particularly sensitive to choice of parameter values, and that validation using experimental data is essential in order to obtain accurate predictions from the mathematical model.

INTRODUCTION

In the continuous casting process, molten steel is continuously delivered to a copper mould through a twin-ported submerged-entry nozzle. Heat is extracted through the mould plates, allowing a thin solid steel shell to develop adjacent to the mould as the steel is continuously withdrawn vertically out the bottom of the mould. The molten core of the casting is subsequently solidified by water sprays below the mould as it travels between guide rolls for a further 30 metres or so.

A steady-state, three-dimensional mathematical model has been developed to describe the turbulent heat transfer, fluid flow and solidification in the mould region of a continuous slab caster, and has previously been described (Flint (1990)). Figure 1 shows typical velocity and temperature fields predicted by the model. It can be seen from this figure that the molten steel exits the delivery nozzle (SEN) in a jet, impacting with the narrow face and then being split into two streams, one which returns just below the meniscus towards the SEN, and the other which travels downwards. This latter stream can be seen to be partially recirculated back towards the SEN from underneath. The resulting temperature distribution shows that the steel loses superheat as it travels around the two recirculation regions just described, the implications of which will not be discussed here.

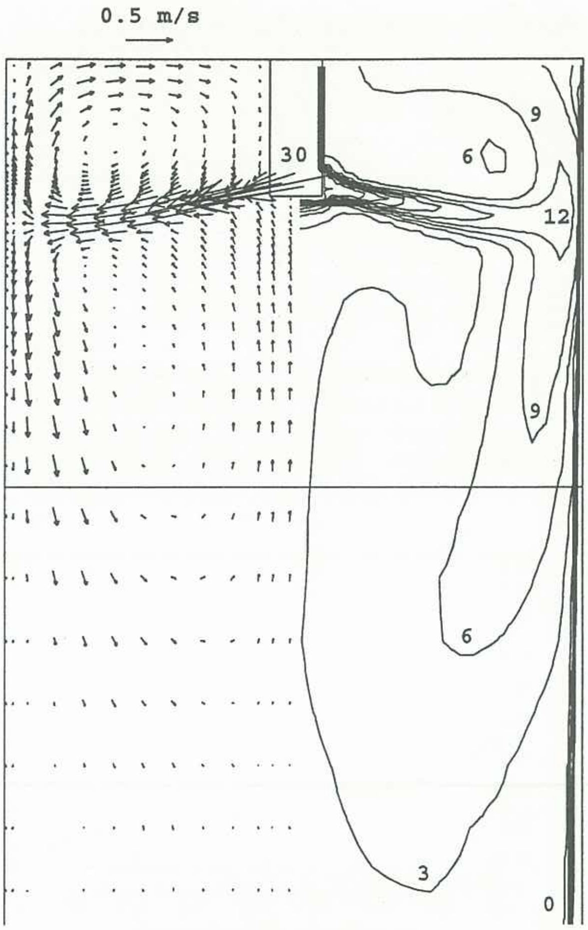

Fig. 1. Mathematically predicted velocity vectors and temperature distribution (shown as superheat in °C) in a 1100mm x 230mm slab caster mould, at a casting speed of 1.5 m/min and a delivered superheat of 30°C. This is the centreline section of a 3-d simulation. Note that not all velocity vectors in this plane are shown, for reasons of clarity.

Before using the developed model for the quantitative prediction of flow and temperature fields, the fluid flow aspects of the model were calibrated using data taken from full-scale water modelling experiments conducted at BHP Research Labs (He (unpublished)). It is this aspect of the research on which this paper will concentrate. Calibration of the molten steel flow model using a full-scale water model is possible because the kinematic viscosities of fully molten steel and water differ by approximately 10%.

MODEL DESCRIPTION

The governing equations for the water model simulation are those of three-dimensional, steady-state, turbulent, incompressible transport of mass and momentum, which may be stated in time-averaged form as follows:

$$\nabla \cdot \mathrm{v} = 0 \tag{1}$$

$$\rho \mathrm{v} \cdot \nabla \mathrm{v} = \mu_E \nabla^2 \mathrm{v} - \nabla p \tag{2}$$

The effective viscosity μ_E in (2) can be expressed as

$$\mu_E = \mu + C_\mu \rho \frac{k^2}{\varepsilon} \tag{3}$$

where C_μ has a value of 0.09, $k(x,y,z)$ is the local turbulence energy, and $\varepsilon(x,y,z)$ is the rate of dissipation of turbulence energy. Both k and ε have their own transport equations, as given by Launder and Spalding (1974), which must be solved along with the equations for mass and momentum transport.

The runs reported in this paper are for a slab caster casting 230 mm x 1100 mm sections. There is assumed to be 800 mm of steel in the mould, and the simulation extends to 2000 mm below the meniscus. SEN submergence depth is 210 mm to the port centrelines, and the casting speed is 1.5 m/min.

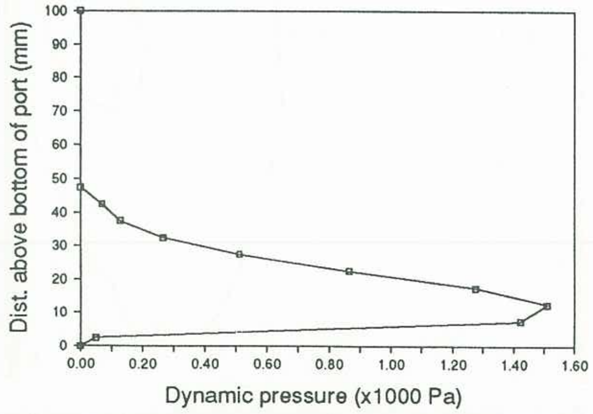

Fig. 2. Measured dynamic pressure vertically down the centreline of a port of a typical SEN (total port height 100mm).

Inlet Boundary Conditions

The fluid is assumed to enter the solution domain at the SEN port exits. A finite difference mesh of typically 8x6 cells cover each port, and in each of these cells, the inlet values of each velocity component, and turbulence quantities k and ε must be specified. The velocity field at the SEN ports is calculated from experimental pitot tube measurements from a full-scale water model. An example of a measured dynamic pressure profile at the SEN ports is shown in Fig. 2. For all runs, inlet values of k and ε are assumed to be constant at the ports, for the lack of experimental information, and are given, except where explicitly stated, by expressions derived from fully-developed pipe flow;

$$k_{\mathrm{in}} = (0.055\overline{v}_{\mathrm{in}})^2 \qquad \text{and} \qquad \varepsilon_{\mathrm{in}} = \frac{C_\mu k_{\mathrm{in}}^{3/2}}{0.03 D_{bore}} \tag{4}$$

where D_{bore} is the SEN bore diameter and $\overline{v}_{\mathrm{in}}$ is the mean inlet velocity through SEN ports.

Equations (1) to (3), and associated boundary conditions, are solved using a finite difference discretisation, derived from the TEACH-T code of Gosman and Ideriah (1976).

MODEL TUNING

Experimental data used for model tuning took the form of forty-eight narrow face pressure tappings (12 rows of four tappings) for each water model run, from which the narrow face impact pressure and position could be infered. For tuning purposes, the mathematical model was run using the properties of water. Several aspects of the model had to be checked and adjusted, and these will now be discussed in turn.

Obtaining a Grid-independent Solution

It is well established that any two- or three-dimensional finite difference discretisation of advection-diffusion equations will introduce "numerical diffusion" into the problem whenever flow skew to the grid mesh is present (Patankar (1980)). It is also known that as a solution mesh is made progressively finer, this false diffusion will be reduced asymptotically to zero. However, as a grid mesh is made finer, CPU requirements increase.

Figure 3 shows the influence of the number of cells in each coordinate direction on the predicted narrow face impact pressure. In all the simulations presented in Fig. 3, the "flux-limited QUICK" discretisation, described below, is used. The mesh chosen for the remaining model validation and tuning work is 17x26x42, which represents a trade-off between solution accuracy and CPU requirements.

Choice of Discretisation Scheme

Choice of discretisation scheme has a considerable influence on the flow field. The popular "hybrid upwind" scheme, known to allow significant numerical diffusion under conditions of flow skew to the grid mesh, predicts a narrow face impact pressure 28% lower than that predicted by the more accurate second order "flux-limited QUICK" scheme (Gaskell and Lau (1988)). This difference is due to the greater artificial diffusion of each SEN jet, in the case of the hybrid upwind scheme, as it travels towards the narrow face. The "flux-limited QUICK" scheme is therefore chosen as the discretisation scheme for all the remaining simulations presented in this paper, despite an approximately 10% greater CPU requirement than the standard hybrid scheme.

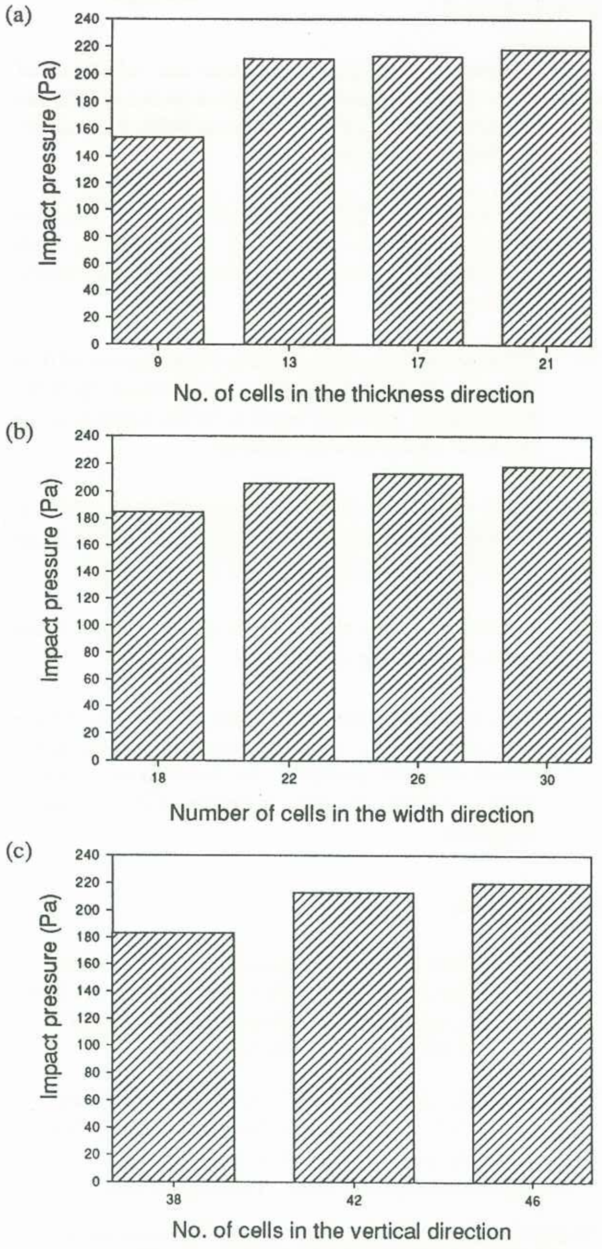

Fig. 3. Effect of mesh density (a) in thickness direction (26x42 cells in other directions), (b) in width direction (17x42 cells in other directions), and (c) in vertical direction (17x26 cells in other directions), on predicted narrow face impact pressure.

Choice of Turbulence Model Parameters

Increasing the $k-\varepsilon$ turbulence model parameter C_2 has the effect of decreasing the rate of turbulent dissipation, and hence increasing the turbulent diffusivity, and this is the cause of the observed 6.5% drop in impact pressure brought about by an 11% increase in the value chosen for C_2 (see Fig. 4).

Choice of Wall Friction Treatment

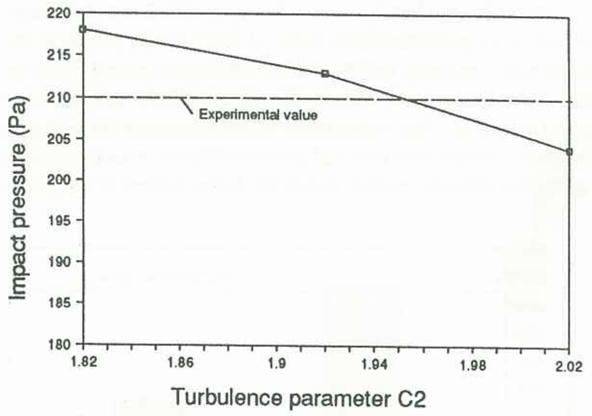

Fig. 4. Effect of $k-\varepsilon$ turbulence model parameter C_2 on predicted narrow face impact pressure.

The manner in which wall friction is handled, by so-called "wall-functions", has a significant effect on the predicted overall flow pattern, and on predicted levels of turbulence near the walls. A wall-function treatment in which the dimensionless wall friction scaling distance y^+ is calculated from the local turbulence intensity (as used in the original TEACH-T code of Gosman and Ideriah (1976)) was found to give a reasonable prediction of wall drag effects. In particular, at the narrow face impact points, this type of treatment correctly predicts a maximum in the local turbulence intensity, whereas a wall-function treatment in which the dimensionless wall distance y^+ is calculated from the local tangential velocity (as supplied as an option in the PHOENICS flow simulation code (1987)), seriously under-predicts turbulence levels in the impact zones.

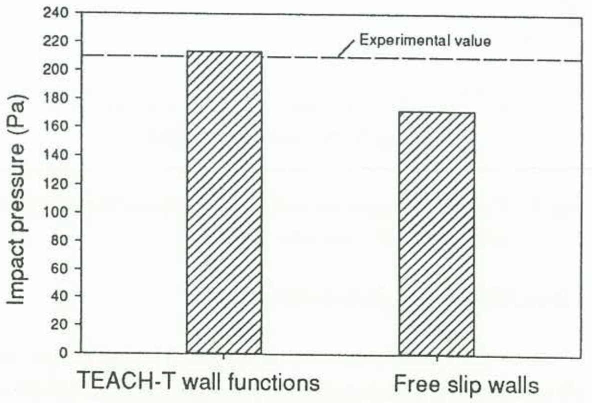

Fig. 5. Impact pressures predicted using "free-slip", and "TEACH-type" wall boundary conditions.

It was also found that using no wall-function treatment (i.e., "free-slip" boundary conditions), leads to a predicted narrow face impact pressure 19% lower than that predicted using wall-functions based on the local turbulence intensity (see Fig. 5).

Choice of SEN Port Velocity Profile

It was found that the inlet velocity profile has a very large effect on the predicted flow field. A flat velocity profile at the SEN ports seriously under-predicts the impact pressure, due to the lower momentum compared with a spatially non-uniform port flow field. Fig. 6 compares predicted narrow face impact pressures for the two cases of a measured port velocity profile, and a flat velocity profile out of the lower halves of the ports.

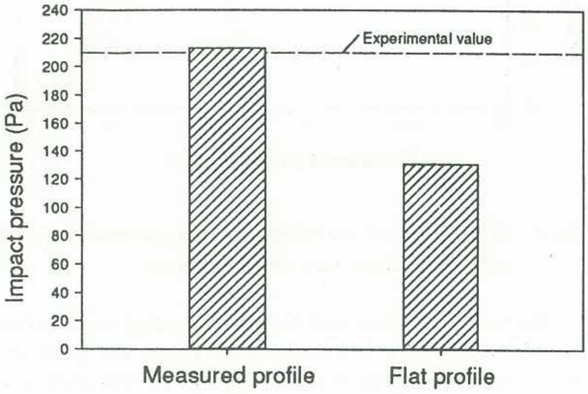

Fig. 6. Impact pressures predicted using a measured, and a flat, velocity profile at the SEN ports.

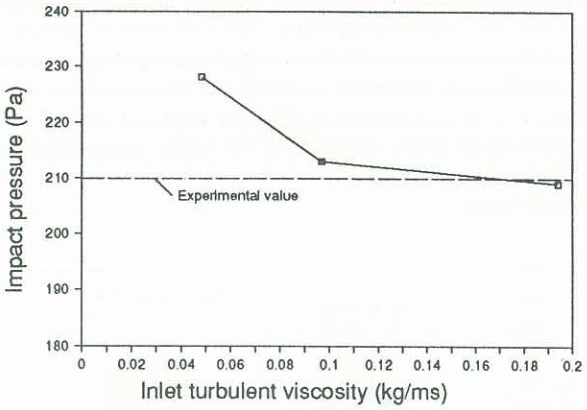

Fig. 7. Predicted impact pressure as a function of the assumed port turbulent viscosity.

Choice of SEN Port Turbulence Level

Inlet values of the turbulence variables k and ε have an effect on SEN stream dispersion rate. Figure 7 shows the effect of varying the assumed value of the port turbulent viscosity on the narrow face impact pressure. The middle point on Fig. 7 corresponds to Eq. (4).

PARAMETRIC STUDIES

The calibrated model has been utilised to increase our understanding of the implications of fluid flow behaviour in the mould from the stand point of shell growth/remelting. Results of parametric studies are reported elsewhere (Flint et al. (1992)).

CONCLUSIONS

This paper has reported the validation and calibration of the fluid flow aspects of a three-dimensional model of turbulent heat and momentum transport in the steel slab caster mould. The major findings are as follows:

(i) Before any matching of numerical and experimental predictions can be carried out, the minimum mesh density which gives mesh-independent solutions should be found.

(ii) In the present case, where there is a large amount of flow skew to the grid mesh, the "flux-limited QUICK" discretisation scheme is found to be far superior to the standard "hybrid upwind" scheme.

(iii) The $k - \varepsilon$ turbulence model is known to be sensitive to the values of the empirical constants in the model, a fact also demonstrated in the present investigation.

(iv) Model predictions are sensitive to assumptions made about wall boundary conditions.

(v) Inlet boundary conditions, at the SEN ports, have a considerable effect on model predictions, and experimental data has proven to be essential in choosing correct values of inlet velocity and turbulence quantities.

REFERENCES

FLINT, PJ (1990), "A three-dimensional finite difference model of heat transfer, fluid flow and solidification in the continuous slab caster", *ISS 73rd Steelmaking Conf. Proc.*, pp. 418-490, Detroit MI, Mar 25-28.

FLINT, PJ, HE, QL, MAHAPATRA, RB and HERBERTSON, J (1992), "Heat transfer, fluid flow and solidification modelling of the continuous casting mould", *ISS 75rd Steelmaking Conf. Proc.*, Toronto, Canada.

GASKELL, PH and LAU, AKC (1988), "Curvature-compensated convective transport: SMART, a new boundedness-preserving transport algorithm", *Int. J. Num. Meth. Fluids* 8, pp. 617-641.

GOSMAN, AD and IDERIAH, FJK (1976), "TEACH-T : A general computer programme for two-dimensional, turbulent recirculating flows", Fluids Section Report, Mech. Eng. Dept., Imperial College, London.

LAUNDER, BE and SPALDING, DB (1974), "The numerical computation of turbulent flow", *Comp. Meth. App. Mech. Eng.* 3, pp. 269-289.

PATANKAR, SV (1980), "Numerical heat transfer and fluid flow", Hemisphere.

PHOENICS flow simulation code, Version 1.4 (1987), CHAM UK Ltd., Bakery House, 40 High St., Wimbledon, SW19 5AU, England.