Key Engineering Materials Vol. 443 (2010) pp 15-20

doi:10.4028/www.scientific.net/KEM.443.15

Modeling the Vertical Spincasting of Large Bimetallic Rolling Mill Rolls

Léo Studer[1,a], Sylvain Detrembleur[1,b], Benjamin J. Dewals[1,2,c], Michel Pirotton[1,d] and Anne Marie Habraken[1,2,e]

[1]Solid, Structural & Fluid Mechanics division (MS2F),
Architecture, Geology, Environment & Construction department (ArGEnCo),
University of Liege, Bat B52/3, Chemin des chevreuils 1, 4000 Liege, Belgium

[2]Belgian Fund for Scientific Research F.R.S.-FNRS

[a]leo.studer@ulg.ac.be, [b]sylvain.detrembleur@ulg.ac.be, [c]b.dewals@ulg.ac.be,
[d]michel.pirotton@ulg.ac.be, [e]anne.habraken@ulg.ac.be

Keywords: Hot rolling mill roll; fluid mechanics; finite volumes method; projection method; conservative level set method; finite elements method; metallurgy; solid mechanics; solvers coupling; semi high speed steel; metal solidification; dynamic effects; spincasting

Abstract. In order to take into account the dynamic effects of molten metal during solidification, a methodology is presented to interface a metal solidification solver (coupled thermal mechanical metallurgical finite elements solver) with a specifically developed flow dynamics solver. (flow dynamics and thermics finite volume solver) The numerical set of tools is designed to be used for the simulation of bimetallic hot rolling mill rolls vertical spincasting. Modeling the industrial process for these products imply certain specifications on the numerical methods used, mainly due to the size of the geometrical domain, low Rossby & Ekman numbers, and a high Reynolds number.

Introduction

The aim of this project is to provide a numerical solution for simulating the centrifugal casting of large bimetallic rolling mill rolls. These products are made up of two different materials: a spheroidal graphite iron as core material, and a high resistant steel as outer material. Both materials are sequentially poured into a spinning mold, the outer material being poured first. In 1970, the Belgian manufacturer Marichal Ketin adapted the vertical spincasting process for bimetallic hot rolling mill rolls manufacturing (see Fig. 1). Since then, this technology has been successfully used, with empirical developments and adjustments.

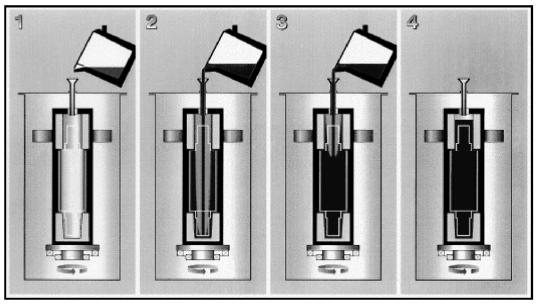

Fig. 1 Industrial process

However, the recent introduction of semi high speed steel (semi HSS) as outer material leads to unpredictable defects, weakening the whole structure of the product. The nature of this casting technique leads to a need for numerical simulation: due to high rotation speed and temperature, visual observation is said to be impossible and sensors use is very restricted. In order to overcome the computational cost of the simulation, a fluid solver and a solid solver are coupled, and a two dimensional approach is used, despite its physical shortcomings.

Global Methodology

The strategy for the developments of this project mainly falls into three categories:

- Building a finite volume method (FVM) fluid solver with complex abilities;
- Characterizing the material parameters that will be used with the finite element method (FEM) thermomechanical metallurgical simulation;
- Interfacing both the FVM fluid solver and the FEM solid solver.

As a result of on-going research, a two dimensional fluid solver is being developed using the classical projection method. Regarding free surface tracking, the level set method has been preferred. Following the spirit of the previous developments by the applied hydraulics team, a conservative formulation of level set is used [5,6]. Actually, code developments include dealing with the issues raised by such a formulation, and the validation based on classical test cases. Another fundamental part of the project is material characterization, required by both fluid and solid solvers. This is fundamental to the solidification simulation. The metallurgical solver requires material parameters to be characterized for each solid phase, and this is often difficult to achieve in practice.

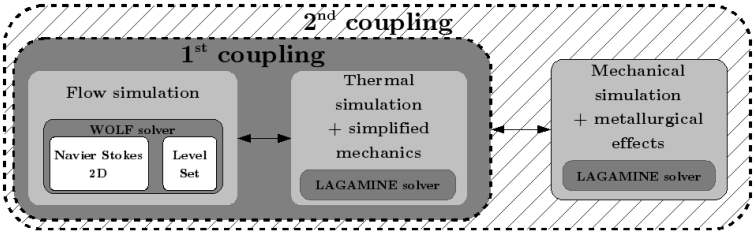

Fig. 2 Global interfacing strategy

The basic assumption of the project is that the fluid simulations will keep on running until the whole liquid metal becomes solid. Three distinct areas are defined: the solid, the liquid and the mushy zones. In the liquid zone, constant physical parameters might be assumed, and liquid metal assumptions are used. In the solid zone, the FEM thermal mechanical solver [1,2] is used. The mushy zone is determinant: here, the physical parameters evolve and lead to solidification, it is also less geometrically and temporally steady than the two other domains. Given an ad-hoc time scale analysis, informations between both simulations are exchanged. In order to avoid the occurrence of too many convergence problems simultaneously, interfacing is done in two steps: the first one neglecting the large complexity of some metallurgical phenomena, and the second one taking them into account (see Fig. 2). A major issue when trying to numerically simulate the vertical spincasting of large hot rolling mill rolls is the extend of the geometrical domain. A three-dimensional model fully integrating fluid and solid equations is reasonably out of reach, at least with a mesh characteristic size sufficient for predicting the apparition of tiny cracks in the structure of the roll.

Solidification Modeling

Thermal Mechanical Metallurgical Solver. The solid model is implemented in the LAGAMINE non-linear finite element code, which has been developed in house [1,2] by the ArGEnCO department since the early eighties. As it is dedicated to solid mechanics modelling, LAGAMINE has been written in updated material (Lagrangian) formalism. No fluid flow is thus modelled: molten steel is only present in the model to provide matter to solidify and apply ferrostatic pressure. The modelling of the thermal metallurgical analysis uses an original differential formulation of the diffusive transformation model and the Koistinen Marburger [4] law for martensitic transformation. The metallurgical model is based on time temperature transformation (TTT) diagrams and assumes the validity of the additivity principle in order to simulate the diffusive phase transformations during any given thermal history. Due to the evolving chemical composition of the semi HSS

matrix during cooling, the initial TTT diagram cannot be expressed through the means of a Kirkaldy-like [3] method. However, inverse analysis of available continuous cooling transformation (CCT) diagrams directs the research team when defining the input. The model computes for each integration point the thermal flows, stress tensors, and percentages of presence for each solid phase: austenite, ferrite, perlite, bainite and martensite. The model can take into account numerous couplings, as shown on Fig. 2. At first, a simplified approach is used, substituting the costly computation of exact metallurgical states by a trivial approach where a variable dilatation coefficient will account for phase transformations. The full model will be used at first in order to adjust the thermal dependence of this coefficient. This solidification model has been extensively compared and validated [1,2]. The mechanical behaviour is implemented using a thermo-elasto-plastic law.

Material Parameters Characterisation. The range of parameters to characterize is quite wide, and can be split into two parts: the thermophysical parameters, and the thermomechanical parameters. The first group includes, for each solid phase: density, thermal conductivity, specific heat, linear thermal expansion coefficient. For each phase transformation: latent heat and strain of transformation. Furthermore, viscosity as a function of temperature, and eventually the heat transfer coefficients of the mould with its surroundings are needed. Amongst the second group, for each solid phase: Young's modulus, Poisson's ratio, yield stress and hardening parameter. Eventually, the big part of this second characterization would be the establishment of the TTT diagram for our steel. Last but not least, the stress effect on each phase transformation has to be expressed. The studied hot rolling mill rolls are constituted by two fundamentally different materials: spheroidal graphite iron in the core, and semi high speed steel in the shell. These have totally different mechanical and physical behaviour, so parameters characterization has to be performed for both alloys. The parameters characterization started [9] with a 6 month experimental campaign in Padova University, where a *Gleeble system* and a *material testing system* (MTS) have been used for mechanical characterization under high temperature loading. Experimental testing will be carried on in parallel with the numerical developments. Current tests include *differential thermal analysis* (DTA), *differential scanning calorimetry* (DSC) and *cryogenic quenching*.

Molten Metal Dynamic Modeling

The solver used for modeling the liquid is called WOLF2DV. This solver is a part of a hydrodynamic modelling system developed for about ten years within the unit of Hydrology, Applied Hydrodynamics and Hydraulic Constructions of the University of Liege. It is based on the technique of the finite volumes applied on a structured grid. Time integration is ensured by the well known Runge-Kutta schemes.

Solving the Navier-Stokes Equations: the Projection Method. In order to model the liquid phase of molten metal, which viscosity and density are experimentally characterized, the Navier-Stokes equations are solved in the vertical plane of the flow. Several resolution methods can be applied such as the implicit method, the method of pseudo-compressibility but also the projection method [7] which was in this case selected. It consists into separating the resolution of the equations into two steps. The first one is called the *transport* step while the second one is the *projection* step. The transport step consists in neglecting the pressure gradient and keeping only the two velocity unknowns. The resolution of the two equations of momentum balance provides a first approximate velocity field, which is not divergence free. Next, the projection step consists in restoring the incompressibility of the velocity field by resolving a Poisson's equation on the pressure with a source term corresponding to the divergence of the approximate velocity field. This step enables to determine the new field of pressure at next time step but also provides the corrections that must be applied to the velocity field to obtain the final field with zero divergence.

Free Surface Tracking: the Level Set Method. After solving the Navier-Stokes equations for the liquid phase, the location of the free surface needs to be updated using the level set [8] technique. For this purpose, a divergence free velocity field needs to be computed in the fictitious air domain. Within a thin layer of air cells close to the interface, the velocities are extrapolated from

those previously computed in the liquid phase. As presented by Sussman [10], the extrapolation is conducted following a four-step procedure:

- For each considered cell in the air, tag the cells in the water that belong to a 5 by 5 stencil centered on the closest cell of the interface.
- Estimate by a least square approach the coefficients of a plane approximating the velocity field computed in the fluid.
- Compute the velocities in the air cell from the fitted plane approximation.
- The extrapolated velocity field must be corrected to restore continuity in the air, by solving a Poisson's equation with as a source term the divergence of the extrapolated velocity field. In addition, boundary conditions for the velocities are prescribed at the interface.

The air cells located further from the interface should also verify the continuity equation but they don't need to be extrapolated from the fluid domain. They progressively spoil the characteristic properties of the level set function, but without affecting the interface tracking. If needed, the level set function is redistanced at regular intervals.

Boundary Conditions. Boundary conditions generally prescribed to solve the Navier-Stokes equations are velocities and pressure. In the present approach, the first ones are prescribed in the transport phase, while pressure boundary conditions are prescribed at the stage of solving the Poisson's equation. Pressure conditions are required both at outflow boundaries and at the time-varying free surface. At the free surface boundary, they are prescribed with a second order accuracy and accounting for the real distance between the boundary and the location of the free surface given by the level set function. Boundary conditions are also applied to the level set function at inlets, based on a geometric linear extrapolation of values existing in the domain.

The validity of this solver was checked on many pressurized and free surface flow benchmarks including, for instance, the flow induced in a sloshing tank [11,12], as shown in Fig. 3.

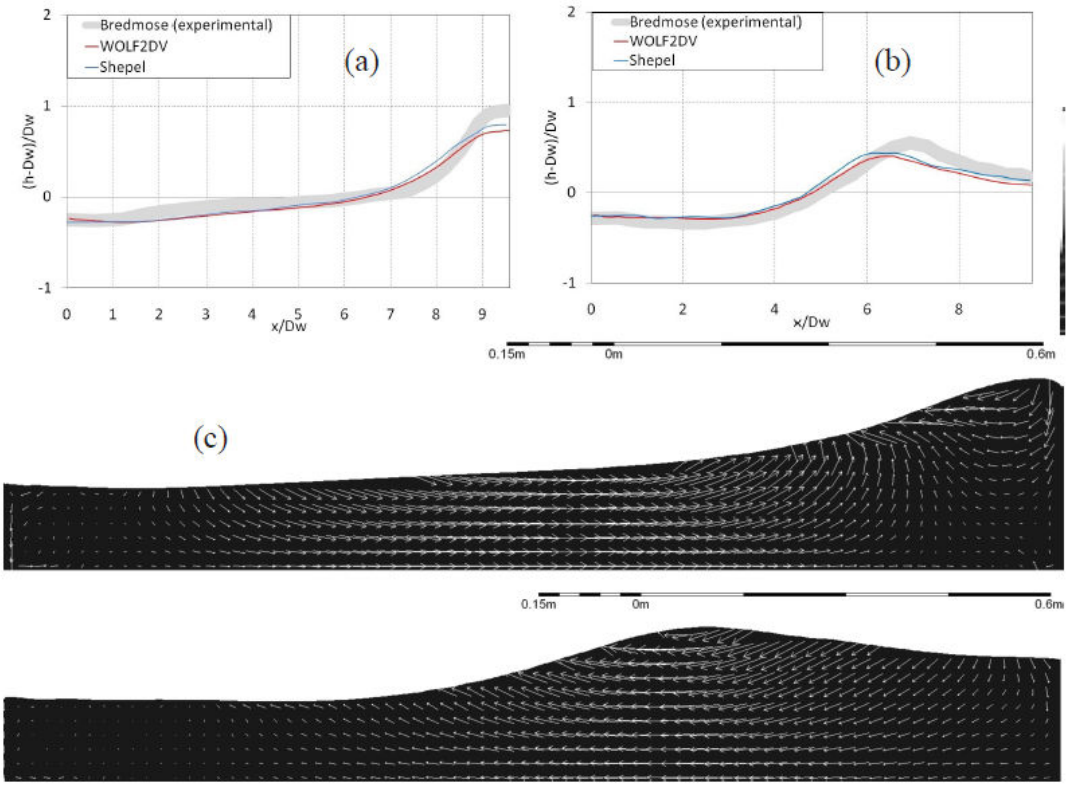

Fig. 3 Computed free surface in comparison with numerical results from [11] and experimental results from [12]. (a) t = 6,52 s., (b) t = 6,88 s., (c) t = 6,52 s., (d) t = 6,88 s. Notation : h is the water depth, Dw the initial water depth and x the abscissa along the tank

Solvers Coupling

Principle. The basic assumption of the project is that we are using two fully independent solvers: one for the liquid phase, computing fluid dynamics and thermics, and the other one for the solid phase, evaluating solidification, solid phase growth, residual stresses, and temperature field within the solid. In order to simulate the flow influence on solid, the FVM solver will give pressure and temperature at the liquid/solid interface, and reciprocally, the FEM solver will provide heat fluxes at the liquid/solid interface. This interface will itself be geometrically updated by the coupling program according to a simple liquidus/solidus rule.

In practice, the solvers have totally independent geometrical domains, and work as two separate entities. Given the fact that we are using a FVM solver, and a FEM one, we must set up a coupling software able to make the correspondence between any interface from a fixed finite volume domain, made up with non-deforming cells, and its counterpart in any "brother" finite elements domain, made up with deformed elements. When the FVM solver sends information to the FEM solver, it is expressed at the center of the cells bordering the solid/liquid interface.

The coupling software must find the corresponding solid/liquid interface in the finite elements domain. Since the corresponding FEM interface has a very low probability to be exactly overlapping the FVM one, the software will also have to interpolate the exchanged information values on the new interface, this time at the edges extremities. (nodes) This interpolated data will then be formatted by the software, and used as an input to the FEM solver next step. Reciprocally, the coupling software will have to interpolate information coming from the FEM nodes towards the FVM edges centers of the solid/liquid interface.

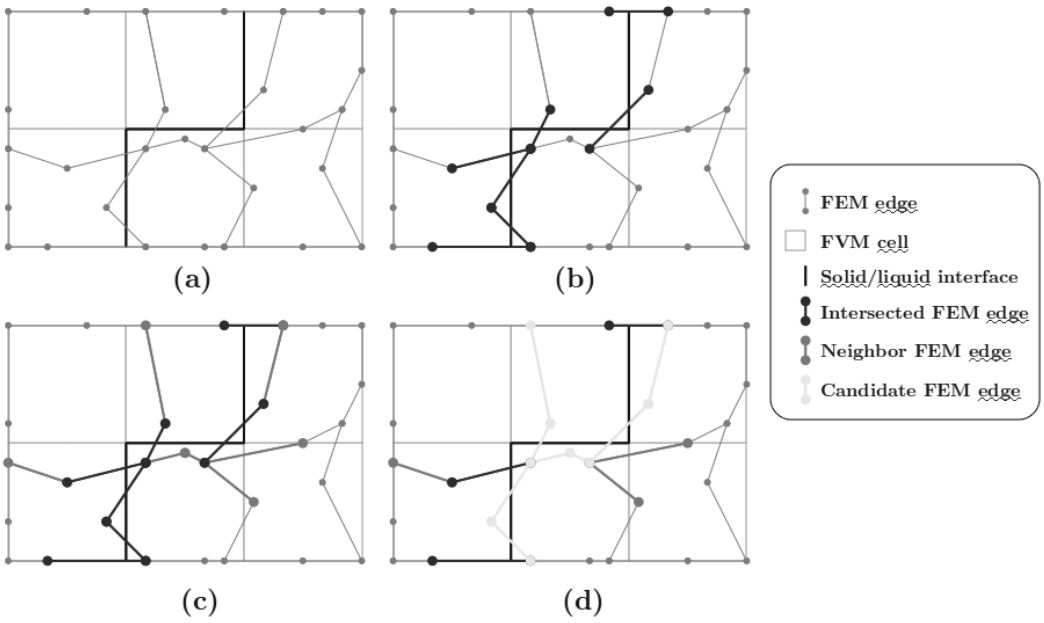

Fig. 4 Finite volumes to finite elements correspondence algorithm

Algorithm. The literature available considering FVM and FEM coupling is scarce, and more specifically with respect to the setting up of an algorithm able to make the correspondence between two interfaces in the respective geometrical domains. We subsequently had to determine an algorithm able to realize this task, as simply and efficiently as possible. As shown in Fig. 4(a) we give ourself a fixed cells FVM domain, an arbitrary solid/liquid interface, and a random FEM domain, random as in geometrically, without consideration for the FEM theory, we are dealing at first with a purely geometrical algorithm that should be able to tackle as much configurations as possible.

The first step for building candidate FEM borders, in Fig. 4(b), consists of determining the FEM edges intersecting the solid/liquid interface. We then build the list of neighboring edges to these

intersected edges, as shown in Fig. 4(c). Eventually we use the set of intersected edges and their neighbors to build candidate interfaces in Fig. 4(d). The candidate interfaces are the sets of connected edges that join the bottom and the upper part of the domain, since we are considering an axisymmetrical problem representing a slice of cylinder. If no candidate border is found in the last step, we iterate step 4(c) and enlarge the set of neighbor edges.

We must then give ourselves a criterion in order to retain only one border in the set of candidates. The desired interface should be the "closer" to the solid/liquid interface. In order to evaluate the "dispersion" of a given border with respect to the reference one, we evaluate the horizontal area separating the vertical edges of the solid/liquid interface and the overlapping edges of a given candidate border, and reciprocally the vertical area separating the horizontal edges of the solid/liquid interface with the overlapping edges of a given candidate border. In the test case figured in this paper, the candidate border that minimizes the "dispersion" with respect to the solid/liquid interface is retained.

References

[1] F. Pascon, S. Cescotto and A.M. Habraken: International Journal for Numerical Methods in Engineering 68 (2006), p.125

[2] A.M. Habraken: *Contribution to the modelling of metal forming by finite element model*, PhD Thesis, Liège University (1989).

[3] J.S. Kirkaldy and B. Burchmays: Journal of heat treatment 8 (1990), p.127

[4] D.P. Koistinen and R.E. Marburger: Acta Metallurgica 7 (1959), p.59

[5] Elin Olsson et. al.: Journal of computational physics 210 (2005), p.225 and 225 (2007), p.785

[6] E. Marchandise and J. Remacle: Journal of computational physics 219 (2006), p.780

[7] A.J. Chorin: Journal of computational physics 2 (1967), p.12

[8] A.J. Sethian: *Level Set Methods and Fast Marching Methods – Evolving interface in Computational geometry*, Cambridge University Press, Berkeley (1999).

[9] L. Studer: *Modelling the vertical spincasting of large bimetallic rolling mills*, DEA Thesis, Liège University (2007).

[10] M. Sussman: Journal of Computational Physics 187 (2003), p.110

[11] Bredmose, H., M. Brocchini, D. H. Peregrine and L. Thais: Journal of Fluid Mechanic, 490 (2003), p. 217

[12] Shepel, S. and B. Smith: Journal of Computational Physics, 218 (2006), p.479