CHAPTER 6: CONCLUSION AND FUTURE WORK

Briefly, this dissertation has highlighted the significant influence of the Submerged Entry Nozzle in the continuous casting process. Consequently, the potential of Mathematical Optimisation of the SEN design has been illustrated quite extensively. Of course, the verification of CFD models using water modelling is a necessity. Only if CFD models deliver reliable and repeatable solutions of different (arbitrary chosen) SEN designs, meaningful optimisation work can be performed based on these CFD results.

A main objective of this dissertation was verifying the CFD models of the SEN and mould with water modelling, using a specifically designed and built 40%-scaled water model. Initially, a purely theoretical optimisation study was expected, being a mere extension from the CFD tundish work (part of the THRIP project at the University of Pretoria), which preceded the SEN and mould work. The complexity and different behaviour of the turbulent jet flow into the mould cavity (as opposed to the mostly laminar and buoyancy-driven flow in tundishes) proved otherwise: extensive CFD model verification was necessary. Trial and error CFD modelling methods (with the aid of "correct" water model results) indicated crucial CFD assumptions, parameters, settings and procedures to ensure repeatable and believable CFD models (Chapter 4). The most critical CFD model parameters or settings were the choice of the correct turbulence model and the quality of the mesh (exclusive hexahedral cells a necessity to minimise CFD errors).

The correctness of the CFD models are measured against water model tests, as the CFD models are verified using water modelling. The reason why correct is written in inverted commas in the paragraph above, is because the water model tests are performed using a 40%-scale water model. Subsequently, experimental design is necessary: three dimensionless numbers have been identified that reflect the specific flow phenomena in the SEN and mould flow. The Fr-number was identified as more important than the Wb-number (which was discarded), second to the most important Re-number. However, an assumption that the flow is independent of Re-number

whilst satisfying Fr-similarity (during water model testing), was proven correct as the results closely corresponded to a full-scale water model in Chapter 3. (A full-scale model simultaneously satisfies Re-similarity and Fr-similarity.)

A further objective was illustrating the optimisation process, focusing on automation of optimisation. Automation in optimisation based on CFD evaluations, necessarily implies that parameterisation in the geometry and mesh is required. This was achieved using the scripting capabilities (ability to interpret text commands sequentially) of the pre-processor GAMBIT. Using the Optimiser (LS-OPT) as the coordinator of the optimisation process, the newly generated mesh geometries from GAMBIT are configured, initialised and solved (according to a predetermined solution procedure) in FLUENT. The optimisation process can be terminated as soon as the objective function (subjected to the constraint functions) has been improved sufficiently.

Lastly, owing to lack of computational power, a 3D design exploration was performed to also illustrate the approximation and global minimisation capabilities of the Optimiser.

However, during the execution of the work described in Chapters 1 to 5, a number of applicable study fields related to this topic, however beyond the scope of this dissertation, were noticed. These fields of study will be reported on in this final Chapter. Moreover, further avenues to explore as an extrapolation on ideas conceived in this work, as well as refinements to certain applications used, are also reported on.

6.1   3D Optimisation

6.1.1   CFD model: further refinements and comments

6.1.1.1 Symmetry assumption

The symmetry assumption used in both 2D (half models) and 3D (quarter models) CFD models proved to be not necessarily true when compared to water model tests. In fact, further work should be performed when one SEN port is clogged more than the other, to evaluate the effect on SEN design performance.

Moreover, the flow in the SEN shaft is not necessarily uniform as assumed, especially when a slide gate is used to control the flow rate through the SEN. This fact causes an asymmetrical flow inside the SEN shaft, which certainly has a significant influence on different jet angles and exit-velocities.

The CFD evaluation of full 3D models is also recommended to investigate the effect of asymmetry in typical plant circumstances, with regards to:

- Viewed from the top of the mould: positioning of SEN inside mould (not in centre of mould)
- Viewed from the side of the mould: angle of SEN with respect to meniscus (not necessarily exactly perpendicular to meniscus)

This topic is also closely related to Robustness studies on optimum designs as predicted by CFD techniques. Refer to section 6.2.

6.1.1.2 Steady / unsteady behaviour of SEN-mould solutions

Unsteady behaviour in some SEN and mould CFD models and water models was observed, especially the models with larger mould widths. Unsteady behaviour was also noticed on SEN designs with small ports and deep wells (refer to Chapter 5 for descriptions). It is believed that the apparent unsteady behaviour is caused by the fact that the flow becomes more complex,

especially in terms of shear flow spreading of the jet that becomes more erratic. Water model tests (refer to Chapter 3 and Appendix F) confirm that a SEN design of the well-type has a more erratic jet spread. CFD results (of the larger width models) also suggest a varying jet angle, oscillating about an apparent equilibrium jet angle.

Although a trial unsteady CFD model (RSM turbulence model) has been solved, using a steady converged solution as the initial solution, not much oscillation was noticed. However, some further work is required as the author suspects that unsteady behaviour takes place in some conspicuous SEN designs (deep well, small ports, large width mould, for example), which complicate the flow.

Furthermore, the choice of turbulence model certainly has a huge impact on the CFD results, as trial and error methods have proven to the author. The less complex the flow, the more capable an inexpensive turbulence model (as the k-ε for 2D flows, and the more advanced k-ω based on Wilcox for 3D flows) proves to be modelling SEN and mould flow situations. The assumption of these inexpensive turbulence models of isotropic turbulence seems to be quite fallacious as flow pattern complexity increases. These choices may have an influence on the steady (or unsteady) nature of a CFD solution.

Moreover, all CFD simulations were forced to yield a steady flow pattern, by assuming that $\dfrac{\partial}{\partial t} = 0$ and $\dfrac{d}{dt} = 0$ (refer to Chapter 2, Literature Survey, for application of these assumptions on the Navier-Stokes Equations). Erratic convergence or even the lack of complete physical convergence (*i.e.*, a physical parameter measured during the iteration or solution procedure that oscillates regardless of residual convergence) may be caused by flow fields that are indeed unsteady (besides the fact that steady behaviour is enforced by the solution algorithm).

6.1.1.3 <u>More refined CFD models (especially on wide moulds)</u>

A full Large Eddy Simulation (LES) model is recommended to be performed, especially for the wider mould widths (1575mm). Using LES modelling, the choice of a turbulence model is irrelevant, as the LES method requires such a fine mesh that a large-scale turbulence model is not necessary – the turbulence variations are computed directly, except for the subgrid scales. Obviously (and unfortunately), LES CFD models are extremely computational expensive. Furthermore, geometric complexity in the SEN design exponentially increases the need for extra-fine meshing.

Currently, however, the resources are lacking for conducting a full LES solution for the base case. However, it is recommended as invaluable future work as soon as an increase in computer power can justify such an exercise.

6.1.1.4 <u>Temperature</u>

The addition of the temperature equation in CFD modelling was required when the real plant circumstances (liquid steel) were modelled (as opposed to imitating the water model where temperature effects are neglected). This fact required additional boundary conditions to be specified on all boundary surfaces with regards to heat transfer. Examples of temperature related boundary conditions are: constant temperature, constant heat flux, varying heat flux, adiabatic, etc.

As specified in Chapter 4 (section 4.5), the constant temperature of the mould walls were specified at the liquidus temperature of liquid steel, as well as a heat flux was specified based on a 1-dimensional study. However, temperatures in the CFD models were not quite accurate – too low temperatures (below liquidus temperature) were obtained in most models.

Therefore, some trial and error work needs to be conducted to fine-tune the heat flux from the mould surfaces to ensure physically correct temperatures.

However, due to the simplifications and assumptions used in the CFD models in this dissertation, temperatures below liquidus temperature will always occur. Figure 6.1 shows the top view of the typical boundary conditions applied to a 3D CFD model. According to the model, the areas in the corners of the meniscus surface are subjected to 3 heat flux extractions, namely those at wide mould walls ($\dot{Q}_1$), narrow mould walls ($\dot{Q}_2$) and meniscus surface ($\dot{Q}_{meniscus}$). This fact causes temperatures below liquidus temperature in these affected areas in the mould corners, which necessarily suggests disastrous meniscus freezing (although plant experience of the base case proves the contrary). The true plant circumstances of course are much more complicated, preventing the unreasonably low temperatures in the corners:

- mould powder in the mould corners prevent excessive heat transfer
- mould oscillation prevents direct contact to walls, also significantly reducing the theoretical heat transfer
- mould powders melt (forming slag) and properties vary, increasing theoretical prediction errors
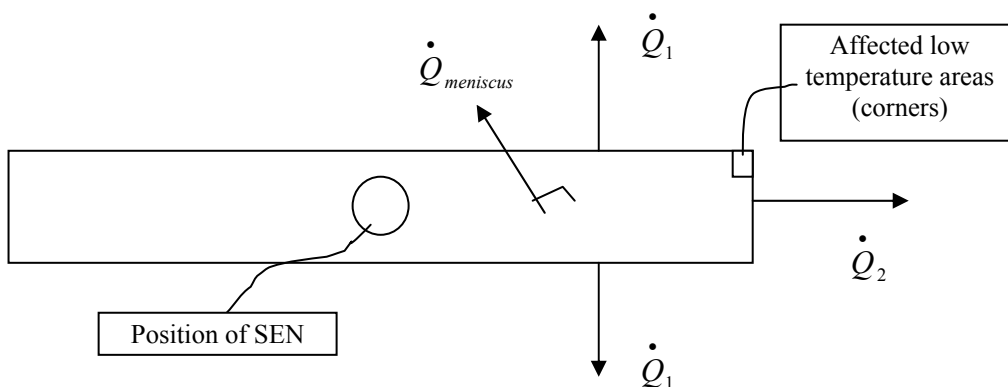

Figure 6.1: Top view of 3D model of SEN and mould, indicating heat flux boundary conditions causing areas of too low temperature

Future work would require extensive study of mould powder properties and behaviour, in an effort to include the (possibly varying) heat resistance of the mould powder (and slag) in the boundary condition of the mould walls and

meniscus surface. The addition of an oscillating mould area must also be considered, as heat flux may also be influenced significantly.

6.1.1.5 Complexity of flow: natural frequency in SEN design and mould widths

Future work is required to exactly ascertain the existence of natural frequencies[1] of a specific SEN design and its influence on flow patterns and meniscus behaviour. This is recommended after a significant increase in maximum turbulent kinetic energy (TKE) was observed when a SEN design was modelled in a wider mould during the 3D design exploration. The same casting speed, SEN design and boundary conditions were used.

The reason why natural frequency might be considered as the culprit for the increase in meniscus TKE for the one specific design, is because throughout the 3D design exploration, most SEN designs showed a decrease in meniscus TKE with an increase in mould width.

A full parametric study (in terms of width variance) should be conducted with a variety of representing SEN designs to evaluate the influence of natural frequency. Variables in this study are predicted to be connected to the specific steel grade (liquid steel density and other properties), mould width, and SEN design (SEN type (welled or not), port height, and port angle).

6.1.1.6 Volume of Fluid (VOF) method for meniscus modelling

Exact meniscus behaviour predictions will become increasingly important as the slag and mould powders need to be modelled for precise plant circumstances imitations. This will (for example) require a Volume of Fluid method of FLUENT to differentiate between three phases (liquid steel, solid

---

[1] The "natural frequency" of a SEN can be defined as certain operational parameters (cast speed, mould thickness, mould width, liquid steel properties, for example) where an unusual unsteady flow pattern occurs within the mould. It is therefore equivalent to the natural frequency of a rotating shaft, where the shaft experiences abnormal vibration and whip at its critical speed (corresponding to its natural frequency).

mould powders and air) over an interface surface. Other free surface methods are also available; refer to Reference [59].

As the exact meniscus behaviour was not important for the CFD simulations in this dissertation (slag entrainment was assumed to be a function of meniscus activity in terms of surface velocity and TKE), the meniscus was physically modelled as a zero shear stress wall. A comparison between a 2D model using an (unsteady) VOF method modelling the meniscus as a free surface, and a model using a slip wall, proved that the flow patterns inside the mould volume are remarkably similar. There had been decided to use the slip wall boundary condition for two reasons:

- a less expensive solution method (steady) can be used, and
- temperature boundary conditions, in particular a heat flux from the meniscus surface, can easily be added.

Currently, using the VOF-method, it will be extremely difficult to specify a heat flux over the free surface, as a heat flux can only be specified on top of the air layer (typically a slip wall), and more uncertainty will be built into this set-up: the heat transfer from the liquid phase to the air, and from the air to the wall, will be unknown. Only the heat extraction from the wall can be specified.

Other methods (than just VOF) must be considered to overcome the heat flux problem. A proposal to consider is to firstly compute the meniscus surface behaviour (wave formation etc.) using a momentum-only CFD model. Thereafter the exact meniscus behaviour (unsteady) must be applied on the meniscus surface, that is dynamically altered (the grid is altered to imitate the exact meniscus surface, yet a slip wall boundary condition is applied) as the unsteady energy activated solution proceeds. Of course, using this proposed method, it is assumed that the addition of heat does not significantly influence the meniscus shape. Some further investigation is thus necessary.

Perhaps other CFD packages with a similar VOF method, yet accommodating easier application of heat flux, can be evaluated and compared with similar FLUENT models (with slip wall boundary) and water model results.

6.1.2   Parameterisation: 3D full optimisation

Due to computational expense, a full 3D parameterisation optimisation study was not possible in this dissertation. The concept (full parameter optimisation) was however extensively illustrated using a 2D SEN optimisation design example.

As computational power increases[2], the possibility of conducting a full 3D optimisation study also increases. There is a need to explore the full implications of 3D geometry in the (arguably simple) SEN designs using parametric studies [25]. Unlike 2D SEN designs, there are a number of influential parameters that are yet to be analysed and screened using the full process described in Chapter 5, sections 5.1 to 5.5. These include the radii of the top ports and bottom ports (which need not be symmetrical), the curvature of the well inside the SEN, to name but a few.

A few design iterations will firstly indicate which parameters are significant (thus, significantly contribute to the improvement (or deterioration) of the objective function(s)). Thereafter, parametric studies and meaningful 3D optimisation can be conducted.

---

[2] Computational power is increasing faster than was anticipated. The average computer employed to perform initial CFD SEN and mould models in this dissertation was an Intel Pentium III 750MHz, 500MB memory. By the time of writing the report, the average system was (equivalent to) an Intel Pentium IV 3.0 to 3.2 GHz, 2GB memory.

6.2   Robustness studies on optimum designs

The robustness of optimum designs will need to be investigated using CFD in a sensitivity analysis. This sensitivity analysis is necessary to ascertain how the optimum design would compensate for manufacturing tolerances (effect on port angle, port height and well depth) as well as operating tolerances (effect on submergence depth and casting speed).

Typical sensitivity analyses require a sample quantity of at least a few thousand to be meaningful. Therefore, instead of performing thousands[3] of CFD evaluations (of different designs imitating typical manufacturing tolerances and operational tolerances), curve fitting through a number of representing optimum design perturbations seems to be the logical approach.

The bounds of variables for the sensitivity analysis will be determined by the manufacturing tolerances. LTM Technologies specified the tolerance on all dimensions as ±1mm, and ±1º for the port angles. Of course operational parameter bounds should also be incorporated in the sensitivity analyses to determine the robustness (or lack of it) of the optimum design in question.

Typically, a sensitivity analysis would compare the objective function of the entire sample block, where each parameter is varied between its expected tolerance bounds. If the objective function value varies significantly for a small parameter deviation (within tolerance), the design will not be regarded as robust. On the other hand, a robust design will show negligible objection function value change for varying (most) parameters within their respective tolerance bounds.

The above explanations on robustness in CFD modelling are (very) brief remarks. Clearly, this subject involves much more detail and work, yet it is anticipated to have a significant impact on the ultimate choice of an "optimum" design. The robustness of

---

[3] Performing thousands of 3D CFD model evaluations will literary take years, even taking into account that computing power will increase following the controversial Mohr's law of computers, which states that average personal computer power will double every two years.

an initial optimum design following an optimisation exercise may be extremely poor; thus, the choice of a poorer (yet more robust) design may well be the better choice.

6.3   Other global approximation methods

6.3.1   Kriging and optimisation with CFD

Kriging can be summarised as a curve fitting method using an interpolation technique between a set of "points". These points are typically similar to the design points explained in Chapter 5, where each point has a certain objective function value as a function of the variables of the design. Only in the case of a 2 variable optimisation exercise these design points can be represented by a 3D graph.

Using the 2 variable optimisation exercise case as an example, a curve can be fitted through a number of these points. Usually, a least squares regression type of fit is used (as used by LS-OPT), to fit a linear or quadratic curve through most of the points. Kriging fits a more accurate curve through these (arbitrary chosen) points, as it relies on a geostatistical approach to modelling. Instead of weighting nearby data points by some power of their inverted distance, Kriging relies on the spatial correlation structure of the data to determine the weighted values. This is a more rigorous approach to modelling, as correlation between data points determines the estimated value at an unsampled point. [Internet source: www.tiem.utk.edu/~sada/help]

Kriging is a powerful tool to be used for optimisation studies, as a more accurate curve will represent the entire design space. Other numerical global optimisation techniques can then be used to minimise the objective function in the domain. Although this method is not necessarily exclusively applicable to the CFD modelling and optimisation of the SEN and mould, it is especially appealing to

CFD optimisation in general, as less CFD model evaluations will be necessary to enable the Kriging surface to represent the entire domain of possible designs.

6.3.2   Neural network approximations

The topic of neural network approximations is well-known and needs no further discussion. These approximations can easily be applied to typical CFD design optimisation studies in an effort to reduce the number of CFD evaluations necessary to perform global optimisation.

6.4   Conclusion

These final remarks on possible future work (refinements to certain applications and further avenues to explore as an extrapolation on ideas conceived in this dissertation) concluded this dissertation.