CHAPTER 2: LITERATURE SURVEY

2.1 Historical Development of Continuous Casting

2.1.1 Historical Background

For well over a century the traditional method for the conversion of liquid steel to solid steel was by use of ingot moulds. Each mould consists of cast iron forming a thick-walled container open at the top and set up before casting on large cast iron *bottom plates* or *stools*. Each ingot was cast independently from a single ladle of liquid steel. There were a number of different ingot mould designs which were mainly divided into the big-end-down moulds and big-end-up moulds [8]. After the liquid steel had solidified in the ingot mould, the mould was removed from the ingot and the ingot was charged into soaking pits (for reheating) for later processing into semi-finished or finished products.

As early as the 19th century, the attraction of solidifying steel in a more continuous fashion was recognised by pioneers as G E Sellars (1840), J Laing (1843) and H Bessemer (1846) [1][8]. These pioneering continuous casting methods were mainly applied to non-ferrous materials with low melting points: it was used for the production of lead tubings and the production of glass [1]. Continuous casting was not applied to steel yet owing to the many technical problems associated with high temperatures involved and the low thermal conductivity of steel.

However, R M Daelen pursued the possibility of solidifying steel using a water-cooled mould, open at the top and bottom in 1887. He patented[1] and envisaged a process comprising:
A stream of liquid steel was poured vertically into an open-ended mould and then passed into a secondary cooling system and withdrawn by pinch rolls prior to being cut by a torch device. This process would be started by the use of a

[1] German Patent No. 51217 of 30 July 1889 (R.M. Daelen).

retractable *dummy bar*.[2] These features are in essence similar to continuous casting machines still in operation today.

Meanwhile, considerable problems occurred due to the sticking of the solidified shell to the water-cooled mould wall until Siegfried Junghans laid the foundations for modern continuous casting. He suggested (and patented) a non-harmonic mould oscillation, which would not influence the heat transfer between strand and mould [1]. In 1933, the first plant for industrial continuous casting of brass, according to the vertical open-ended mould, was built in Germany by S Junghans.

It was not until the Second World War that semi-industrial pilot plants began to emerge for the continuous casting of steel.

The first pilot plants for the continuous casting of steel were built at Babcock and Wilcocks (USA), Low Moor (Great Britain) and Steel Tube Works, Amagasaki (Japan) in 1946 and 1947.

From 1950 onwards the development of the continuous casting of steel on a large scale developed rapidly. Technological advances, which are applicable to this study, will briefly be mentioned: (More detailed information on the advances (and corresponding dates) can be viewed in References [1] and [8].)

- 1952: German patent by O. Schaeber describing the casting of a bent vertical strand instead of a straight vertical strand
- 1952: The first electromagnetic stirrer designed for continuous casting at Mannesmann by Junghans and Schaeber
- 1956: At Barrow (Great Britain) vertical cutting of the billet strand was replaced by horizontal cutting which implied that the withdrawed strand was bent before cutting

[2] The start-up of the continuous casting process requires a bar head (*dummy bar*), which is marginally smaller in cross section than the mould, to be driven in to the bottom of the mould by steering it up from the bottom of the machine using a dummy bar chain. When liquid steel enters the mould, it solidifies around the claw shaped dummy bar. As soon as the mould is filled with molten steel, the dummy bar is withdrawn and the continuous casting process commences. The dummy bar is then removed from the solidifying strand and parked away from the strand.

- 1963: At Dillinger Steelworks (Germany) the first vertical type slab machine made use of horizontal discharge by bending the slab strand
- 1970 – 1983:
    - Rapid ladle and tundish changing equipment to improve productivity and yield
    - Variable width adjusting moulds to minimise mould changing and thus improve yield
    - Mist cooling using air atomised water to improve cooling efficiency and homogeneity
    - Total shrouding of metal streams from ladle to tundish and from tundish to mould to avoid contact with air (oxygen) in order to improve quality. Shrouding from the tundish to the mould is in the form of a refractory tube and generally known as the Submerged Entry Nozzle (SEN). With the casting of billets, an inert gas shrouding is used (as opposed to a refractory tube) due to the small cross sectional area of the mould opening.
    - Integrated computer control of the complete casting process

2.1.2 Evolution of Continuous Casting Machine Design

The basic principle of the continuous casting process for steel (as envisaged by R M Daelen in 1887) is based on the pouring of liquid steel vertically into a water-cooled copper mould, which is open at the bottom. Heat transfer to the copper mould immediately solidifies the liquid steel and a solid skin (commonly known as the *shell*) is formed which increases in thickness down the length of the copper mould. To avoid sticking of the shell to the copper mould, the mould is reciprocated sinusoidally and a lubricant has to be provided to be an interface between the shell and the copper mould. This lubricant is usually introduced as a casting or mould powder, which melts to form a slag. The slag infiltrates the gaps between the steel shell and copper at the meniscus to provide lubrication [15][16].

The early continuous casting machines were totally vertical: they required considerable height to achieve reasonable production rates per strand. Moreover, the support rolls and pinch rolls beneath the mould were under severe stress due to the ferrostatic forces in the strand. Since approximately 1965, continuous casting machines evolved from totally vertical to the curved type. Refer to Figure 2.1 for the diagrammatic depiction of the principle types of (vertical) continuous caster machines. In recent years the curved mould machine (curved mould with straightening or CS as depicted in Figure 2.1) has been widely used. Multi-radius machines (or rather curved mould with progressive straightening or CPS) are also in use currently, which enable an even further reduction in height and thus ferrostatic forces.

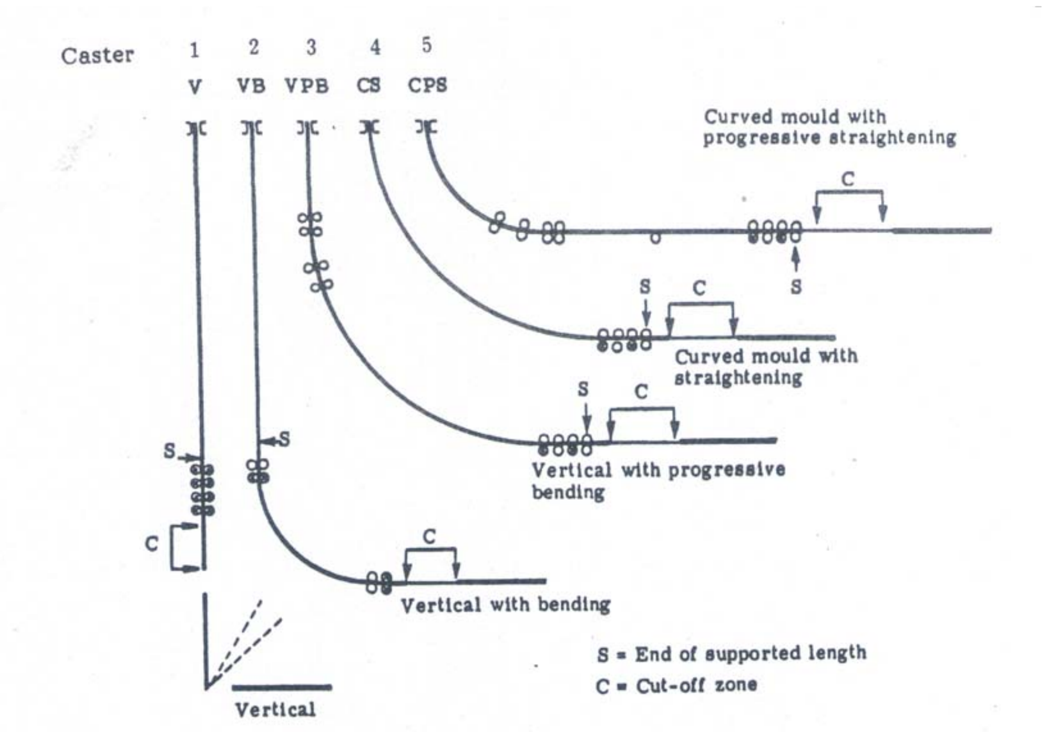

Figure 2.1: Principle types of continuous casting machines [8]

The main advantage of the curved vertical continuous casters is the reduction of machine height with the following benefits:

- reduced costs for plant buildings (lower buildings);
- reduced crane costs (crane height reduced);
- less maintenance (roller gap geometries and roller alignments) for roller support system due to lower ferrostatic forces; and

- no mechanism required to turn the cut off vertical slab horizontally.

The evolution of machine design for slab, bloom and billet casters is depicted in Figure 2.2, where the systematic switch to curved vertical continuous casting machines is emphasised.

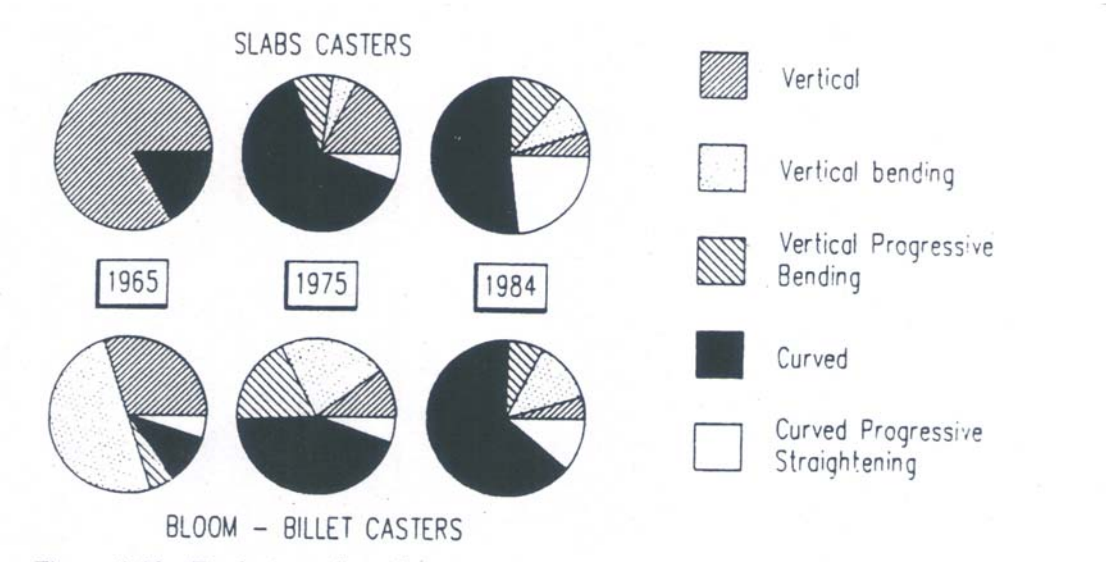

Figure 2.2: Evolution of continuous casting machine design [8]

However, multi-radius machines (Caster 5 (CPS) in Figure 2.1) are limited to a minimum height due to quality difficulties and mould teeming difficulties.

Moreover, by striving to reduce the height of continuous casting machines, in the limit the strand could become totally horizontal. However, considerable difficulties occur with the liquid steel feed arrangement in a horizontal set-up.

2.1.3 Vertical Continuous Casting versus Horizontal Continuous Casting

In the limit where the strand becomes horizontal in order to minimise the machine height, the process is no longer vertical continuous casting, but horizontal continuous casting. As mentioned in the previous section, although the reduction in height implies much less ferrostatic forces and thus simplifying strand support requirements, the steel feed arrangement becomes involved and proves to be difficult. Considerable work has been carried out over the years to further develop total horizontal casting. There are several horizontal continuous casting machines that exist today; however, these machines are mainly limited to billet casters (castings of small cross sectional area – refer to Chapter 1).

As indicated in the previous section, conventional continuous casting can be regarded as vertical casting, which progressed from total vertical casting (maximum height) to the low head / multipoint straightening design (minimum height for conventional casting). Refer to Figure 2.1 in the previous section for the diagrammatical difference between these conventional casting extremities.

However, horizontal casting requires a horizontal tundish-mould joint and special conditions to reduce mould friction owing to the fact that the mould is rigidly fixed to the tundish by means of the feeding link. The mould-tundish link or connection is made of refractory material, which is called the break ring [1][8][17].

During casting, the mould-tundish connection (henceforth break ring) remains fixed and the solidification process is controlled by the withdrawal machine with phases of pull and pause.

Figure 2.3 shows a typical horizontal caster with stationary mould and moveable tundish in casting position. The connection between the tundish and mould (which has a similar function as the SEN) is schematically shown in Figure 2.4. The typical withdrawal cycle of a slab of horizontally casted steel comprises equal phases of pull and pause.

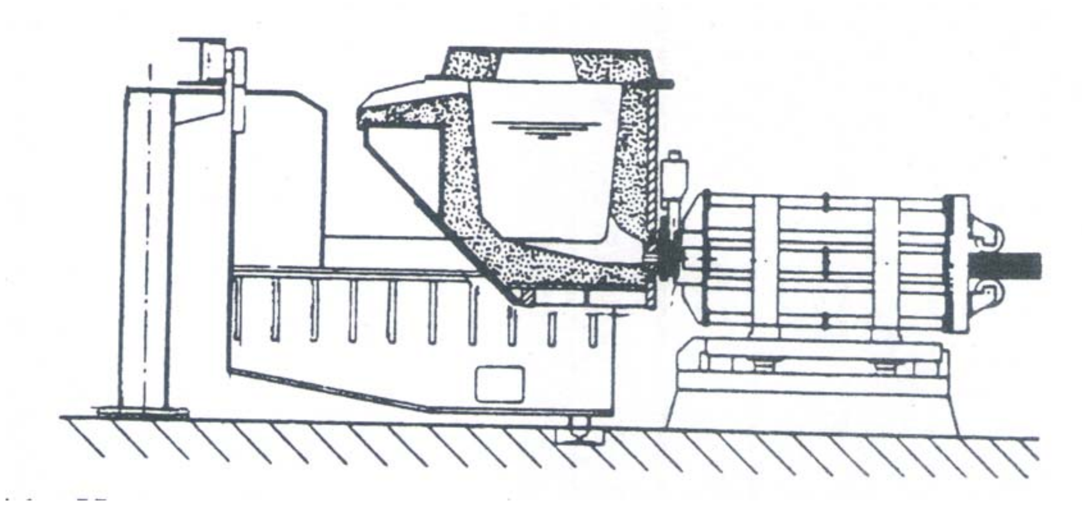

Figure 2.3: Horizontal caster with stationary mould and movable tundish in casting position

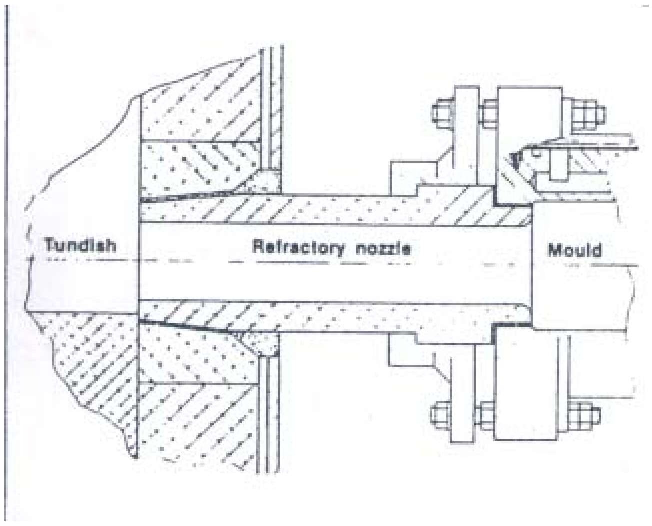

Figure 2.4: Connection mode between tundish and mould, through a refractory nozzle

However, this study involves the mathematical optimisation of the Submerged Entry Nozzle (SEN) in the continuous casting process. Although not stated in the title, this study refers to the optimisation of the SEN in the **conventional** continuous casting process. Quite obviously, due to the absence of the SEN (and the influence of the SEN on the flow pattern inside the mould) in the horizontal casting process, this study is not applicable to the horizontal continuous casting process(es).

2.2 Submerged Entry Nozzle (SEN) Literature

2.2.1 Current continuous casting: background

The current continuous casting process is very similar to that developed by the pioneers in the nineteenth century described in Section 2.1 of this chapter. After the steel has been "mixed" (ingredients or supplements added to molten iron), it is poured into the ladle (Refer to Figure 2.5(a)). The molten steel is then transferred to the tundish, which was traditionally only applied as a reservoir to sustain continuous casting while changing ladles. However, later it was realised that the tundish can also be utilised as a steel purifying vessel. This is achieved by forcing certain flow patterns in the tundish to help extract inclusions and other unwanted particles by entraining the latter in the slag layer on the tundish meniscus [18][19][20][21][22].

The SEN and mould can be regarded as the last casting equipment in the continuous casting process. All other processes afterwards are mainly concerned with extracting the quasi-solidified slab from the mould for further cooling and ultimately to be cut up in slabs for milling (for example).

The direct influence of the SEN on the flow field in the mould will be elaborated on next.

2.2.2 SEN influence on steel

As indicated in previous studies [2][3][4][5][6], the SEN has a primary influence on the flow pattern in the mould and the resultant steel quality: it controls the speed, direction and other characteristics of the steel jet entering the mould. The

SEN also has a major influence on the meniscus behaviour, which has a direct influence on the steel quality.

Figure 2.5 (a) depicts the SEN in the continuous casting process. Figure 2.5 (b) shows (schematically using a 2 dimensional half model of a SEN) the typical parameters that can have an influence on the steel jet exiting into the mould cavity.

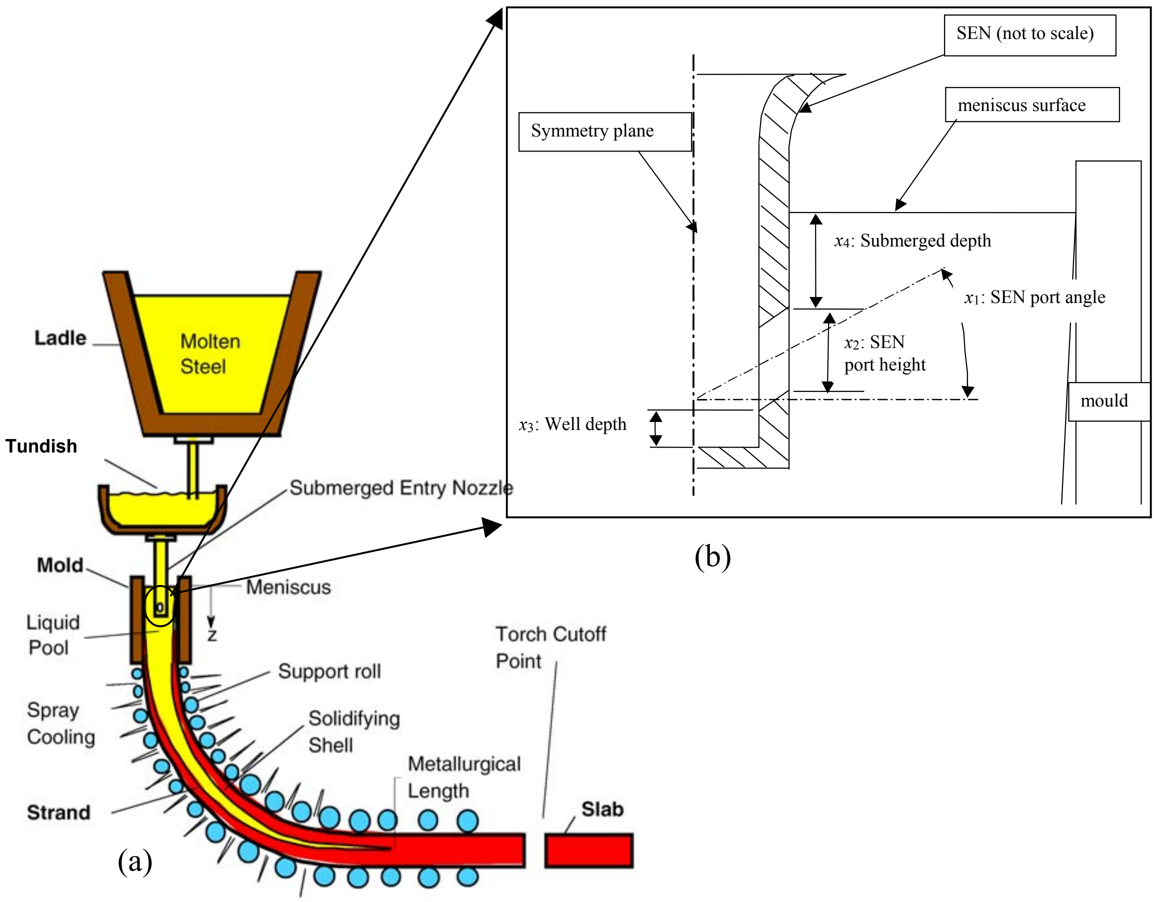

Figure 2.5: SEN in the current continuous casting process and typical influential parameters [7]

Moreover, the SEN is, compared to the complexity of the rest of the continuous casting machinery, rather simple and thus a relatively inexpensive part to change or alter after an optimisation exercise. Consequently, the SEN is an attractive optimisation topic, which will be exploited in this dissertation.

2.2.3 Classification of Literature

The literature referred to in this dissertation have been classified into certain groups. As this dissertation is part of a bigger and ongoing continuous casting research enterprise, the classification of all the references used by the University of Pretoria during the last 4 years will be shown for completeness. Diagram 2.1 depicts most references used during the past four years. Most SEN and mould references are also the references for this dissertation. (The references for the tundish, ladle and inclusion work will be shown in Appendix A for the sake of completeness.)

Diagram 2.1 provides a way of classifying continuous casting literature in major and minor categories. For the references, suffixes are used corresponding to the broad categories:
[ ] = Mould[3], T = Tundish, I = Inclusions, L = Ladle.

Acronyms used in the diagram:
PIV = Particle Image Velocimetry
LDV = Laser Doppler Velocimetry
LES = Large Eddy Simulation
RTD = Residence Time Distribution.

[3] Mould and SEN related references are mostly references for this dissertation.

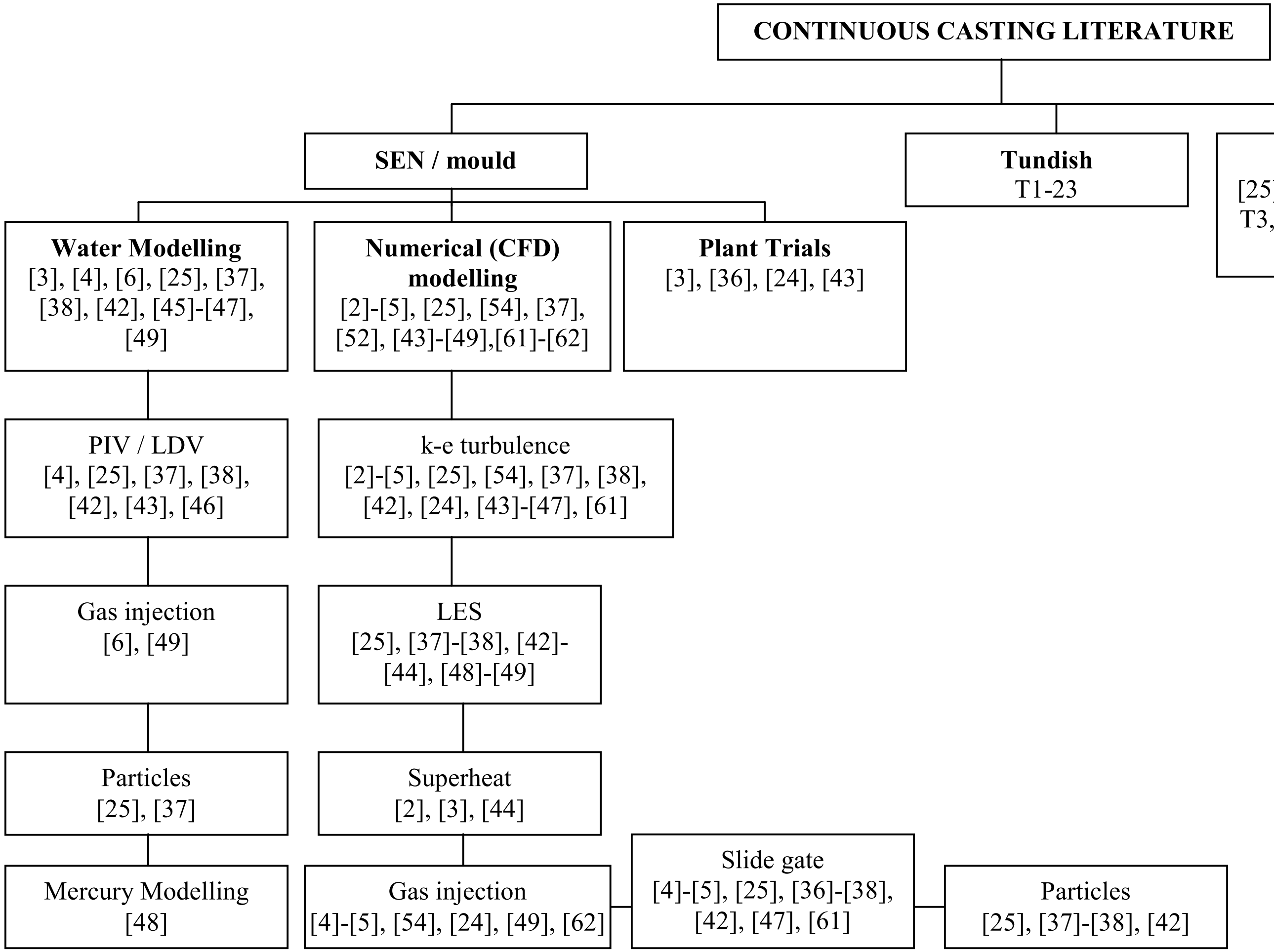

Diagram 2.1: Continuous casting Literature classification

As indicated in Diagram 2.1, the literature on specifically the SEN and mould in continuous casting can be subdivided into three categories:

- Water modelling
- Numerical modelling
- Plant trials

A number of tundish references proved to be quite contributing owing to the similarities in typical problem approaches. CFD[4] models are also validated using water modelling, instead of using traditional plant trial methods. The very similar classification of typical tundish references is shown in Appendix A.

More detail on literature will be discussed in the next section, where previous work on the SEN (and mould) will be mentioned and discussed.

2.2.4 Previous work on Submerged Entry Nozzle design

2.2.4.1 Plant trials

Most plant trials were performed if circumstances were impractical to perform water model tests: e.g., the effects of temperature and surface tension needed to be established. Plant trials were thus very common until approximately a decade ago, where computational models could replace costly plant trials. [23][24].

Plant trials can be very costly, especially if a desired result is not achieved. Moreover, using trial and error plant trial methods, a few unsuccessful iterations can be quite devastating to any steel plant.

Recently, with the global steel price being set by major steel manufacturers, the ever-continuous improvement (or rather cost saving) programmes at steel plants worldwide, prohibits plant trials to take place. Furthermore, high-risk

[4] "CFD" is the acronym for Computational Fluid Dynamics and encompasses the entire study field of Fluid Mechanics using computational or numerical methods.

plant trials influence the efficiency of a steel plant, let alone the possible losses associated with failed (or partially failed) plant trials.

The same trend is followed with tundish design work, where different tundish designs and furniture arrangements are increasingly experimented using CFD modelling and water model verification exclusively. Refer to Appendix A for some tundish references, where striking similarities with SEN design research were identified.

2.2.4.2 Water modelling

Where possible, water model tests were and are mostly performed on full-scale water models of the SEN and mould layout. Owing to the approximate dynamic similarity between water and liquid steel, water models are mostly utilised in an effort to optimise SEN and mould set-ups by acquiring certain desired flow situations for various applications [23]. Water models can however not accurately predict the effect of Ar-bubbles on steel flow, as the relative difference in density is quite marked. The surface tension of liquid steel also differs significantly from the full-scale water model counterparts; subsequently plant trials were a necessity in some cases.

However, since the possibility of numerically solving similar flow situations using CFD techniques (refer to sections 2.2.4.3 and 2.3) with the arrival of powerful enough computers, plant trials are not a necessity during the initial development of continuous casting components.

Although complex numerical models can accurately predict the flow of liquid steel in the SEN and mould with more information available than physical plant trials, water modelling is definitely not obsolete. Water modelling is currently used to verify numerical and/or CFD models, to ensure that subsequent solutions of flow fields are believable and a representation of physical flow. Most previous studies utilised water models (full-scale and smaller scale) to verify CFD models before the CFD solutions are accepted as true and accurate [3][6][18][19][25][26].

Tundish work in water modelling was quite in abundance in the literature. A number of visualisation techniques employed in the water modelling of tundishes can be applied directly to SEN and mould work: PIV (particle image velocimetry) and LDV (laser Doppler velocimetry). The concept of residence time distribution can also be used with SEN and mould water modelling to determine the efficiency of the SEN to remove potential particles with recirculation zones. Refer to Appendix A for further tundish and inclusion references.

2.2.4.3 Numerical modelling of SEN and mould design

Early numerical work

Early numerical modelling of the SEN and mould is distinguished from CFD modelling: early numerical modelling employed analytical differential equations with macro boundary conditions applicable to very specific SEN and mould problems.

These equations are then solved using numerical computational methods[5] developed in the 1970's [9]. These methods were extremely tedious and the complex flow of a jet exiting into a bigger cavity proved to be practically impossible to solve using these early methods.

Specific mathematical modelling (to be solved numerically) of the SEN and mould have been applied in the 1970s and 1980s [9] (as an example) to predict the temperature field and shell profile in the solidifying steel strand as a function of variables such as section size, casting speed and external cooling conditions. These pioneering methods were very toiling as the models had to be set up for a specific case (geometry, flow situation, flow assumptions, amongst others). However, these early numerical models are based on exactly

[5]Example of different numerical methods used to solve differential equations for simple problem specific flows (**laminar** boundary flow in this case):
Crank Nicolson (1947) method, implemented for boundary layers by Blottner (1970); and the "finite volume" method pioneered by Patankar and Spalding (1970) – implemented in the computer code GENMIX by Spalding (1977).

the same flow equations (more detail in Section 2.3) as that of current CFD codes, and the results of early work certainly paved the way for later (i.e., current) computational work.

CFD modelling work

Current commercially available CFD techniques can be applied to any geometry and any flow situation. Although much more computing power is required than problem specific computer solutions, the solution of complex flow phenomena are now available to the general engineering public, and not only restricted to mathematicians who are able to manipulate problem specific differential equations for numerical solutions. Section 2.3 that follows shortly will briefly explain Computational Fluid Dynamic techniques.

Much work has been done regarding SEN design using numerical modelling methods. Refer to Diagram 2.1 in this Chapter for all SEN/mould work references. These references, especially Refs. [2], [4], [5], [61] and [62], laid the foundation for this dissertation, pointing out the effects of nozzle design (e.g. port angle, port size, port geometries, nozzle bottom) and operational parameters (e.g. casting speed, Argon gas injection rate[6], clogging) on the resultant steel quality.

Most work was performed on SEN-mould models using slide-gate valves (between the tundish and SEN) to control the flow rate for a specific casting speed. The slide-gate orientation invariably causes uneven flow distribution through the bifurcated nozzles [5], resulting in asymmetry in the mould with associated quality problems. However, the work in this dissertation is based on a continuous casting set-up making use of a stopper (rod), actuated from above the tundish. Accordingly, the implicit assumption that flow is uniform in the SEN as the flow enters through the annular inlet, holds throughout this dissertation. This assumption (and reality of a stopper-type flow control system) is therefore a simplification of slide-gate continuous casting plants.

[6] Using Ar-gas during continuous casting is beyond the scope of this dissertation.

Regarding SEN design in an effort to obtain quality continuous cast steel, the following information from Refs. [5], [60] and [61] (amongst others) laid the foundation for the optimisation work in this dissertation. The effects of different SEN designs (summarised below) guided the author in selecting meaningful design variables, objective functions, constraint functions as well as sensible design variable bounds.

Summary of typical numerical SEN design work:[7]

Port angle:

The port angle has a major influence on the mean jet angle, which is critical for flow inside the mould. It is interesting to note that the mean jet angle is always more downward than the port angle [61], which was unexpected by the author, but obvious if one considers the downward momentum of the molten steel inside the SEN shaft (before exiting the nozzle ports). It is also noteworthy the turbulent intensity of the mean jet increases with increased angle (positive or negative), implicating that certain bounds should be specified for optimisation work.

Port height:

Increasing the height of a nozzle port (keeping the width constant), implicates the increase of the port area. Defining a *port-to-bore ratio* based on the areas of the two ports and SEN bore respectively, Honeyands *et al.* (Reference [16] in [61]) correlated the area fraction β ([61]) to be 1 with a port-to-bore ratio of 1 (*i.e.*, no recirculation area at the top of the port). By increasing the height of the ports, β is decreased (implicating a larger recirculation area, that may be vulnerable to detrimental inclusion build-up and clogging of the nozzle ports).

Port thickness:

Thicker ports (thus longer ports) tend to shape the mean jet angle more closely to the port walls, increasing the effect of the SEN design on the resulting molten steel jet.

---

7 Based on work done in these references: [2][3][4][5][7][24][36][45][46][61][62]

Port width:

A narrower port will also increase the effect of the port angle and shape, provided that the bore-to-port ratio is larger than 1. If this ratio is smaller than 1, the effect (on the characteristics of the jet) of the width is less than that of the port height.

For this reason the width has been kept constant for optimisation work in this dissertation.

Port shape:

Round ports increase the swirling component of the jet relative to square ports. This may lead to increased spread angles, increased turbulence intensity and higher ineffective area fractions.

Bottom design:

The bottom design (which may be either flat, recessed or in-line with the bottom port angle) seems to have influences on the meniscus behaviour and general turbulence intensity in the mean jet area.

Influences of varying casting speed:

By increasing the casting speed, the jet speed and turbulence levels merely increase. Strangely, it does not affect the jet angle or other characteristics (recirculating area, spread, amongst others) of the jet.

Other insights:

The flow through the nozzles and into the mould cavity was regarded as steady turbulent flow in mostly all references. This assumption is also incorporated in all work in this dissertation study.

CFD work on other related casting equipment

Not surprisingly, more coinciding CFD techniques exist between the tundish and SEN/mould work. However, a major difference between the two continuous casting subjects is the fact that the flow can be assumed to be laminar in a tundish, as opposed to fully turbulent flow through the SEN and as the jet exits into the mould cavity. Of course this fact has implications on the choice of turbulence models during the CFD modelling process. Refer to Appendix A for these references.

2.3 CFD background

2.3.1 General: Numerical modelling and CFD

2.3.1.1 Introduction: basic equations

Currently, the mention of "CFD" is synonymous with commercial CFD packages such as FLUENT [10], CFX, STAR-CD and MSC Flow to mention but a few.

However, as already defined in footnote 4, "CFD" is the acronym for Computational Fluid Dynamics and encompasses the entire study field of Fluid Mechanics using computational or numerical methods.

It is however interesting to note that all these commercially available CFD packages are built upon the past 5 decades of research in numerical flow modelling [27]. As CFD researchers discover new applications and as computational power increases, commercial CFD packages include these new methods in their programmes in the form of more options.

The basic differential equations on which all CFD packages are built will be briefly presented in this section and the use of computers to solve these equations will be made relevant.

The basic equations are based on the three laws of conservation for a physical system: [9]

1. Conservation of mass (continuity)

2. Conservation of momentum (Newton's second law)
3. Conservation of energy (first law of thermodynamics)

The three unknowns, which must be simultaneously derived from these three basic equations, are the velocity ***v***, the thermodynamic pressure *p*, and the absolute temperature *T*. The final forms of conservations equations (which will be presented shortly) contain four other thermodynamic properties or variables: density ρ, enthalpy *h*, and the two transport properties μ (viscosity) and *k* (conduction). However, these four additional variables are assumed (following the assumption of local thermodynamic equilibrium) to be determined by the only independent variables *p* and *T*.

In order to specify a particular problem completely, the conditions (of various types) for ***v***, *p* and *T* must be known at every point of the boundary of the flow regime. The preceding considerations however apply only to a fluid of uniform, homogeneous composition: *i.e.*, diffusion and chemical reactions are not considered. Multi-component reacting fluids must consider at least two additional basic relations:

4. Conservation of species
5. Laws of chemical reactions

plus additional auxiliary relations such as knowledge of the diffusion coefficients $D = D(p,T)$, chemical-equilibrium constants, reaction rates, and heats of formation.

However, for the purposes of this introduction and basic background to CFD methods, only the differential equations[8] derived from the basic three laws for physical flow will be presented.

[8] To be more precise, partial differential equations (PDEs) are derived from these three basic laws. As it is not the purpose of this dissertation to derive the basic partial differential equations on which the CFD methods are based, these basic equations will only be shown (in basic form). However, whenever these basic equations are "modified" [28] in this dissertation using the FLUENT code [10] to enhance the numerical approximations of the analytical equations, it will be mentioned in the text and indicated accordingly.
In the event that the reader may require the derivations of these basic equations, refer to the following CFD sources in the references: [9][28][29]

The following partial differential equations were derived for general control volumes, expressed in Cartesian coordinates[9]:

1. Conservation of mass: the equation of continuity

$$\frac{D\rho}{Dt} + \rho.divV = 0 \qquad \text{with } \frac{D}{Dt} = \frac{\partial}{\partial t} + (V \bullet \nabla)$$

$$\text{and } divV = \nabla \bullet V = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \qquad \textbf{[eqs 2-1]}$$

where: ***V*** = velocity vector (column)

ρ = density

x, y, z = space coordinates in 3D

u, v, w = velocity components

t = time

2. Conservation of momentum: the Navier-Stokes[10] equations

In scalar form, the Navier-Stokes equations, with the assumption of a general linear (Newtonian) viscous fluid, are presented as follows:

$$\rho\frac{Du}{Dt} = pg_x - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(2\mu\frac{\partial u}{\partial x} + \lambda divV\right) + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right] + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right]$$

[eq 2-2-1]

$$\rho\frac{Dv}{Dt} = pg_y - \frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + \frac{\partial}{\partial y}\left(2\mu\frac{\partial v}{\partial y} + \lambda divV\right) + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)\right]$$

[eq 2-2-2]

$$\rho\frac{Dw}{Dt} = pg_z - \frac{\partial p}{\partial z} + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)\right] + \frac{\partial}{\partial z}\left(2\mu\frac{\partial w}{\partial z} + \lambda divV\right)$$

[eq 2-2-3]

where: in addition to the variables defined in equations 2-1 (above),

[9] These equations can also be expressed in Polar coordinates or Cylindrical coordinates to suit these specific geometries. Refer to [9].

[10] Although the momentum equations are derived from Newton's second law (***F***=***ma***), these equations are known as the Navier-Stokes equations owing to the fact that these equations were only derived following important and necessary assumptions made by Navier (1823) and Stokes (1845). [9]

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + (V \bullet \nabla)$$

μ = viscosity for a Newtonian fluid

3. Conservation of energy: the energy equation (first law of thermodynamics)

With the assumption that the heat transfer to the element volume is governed by Fourier's law[11], the energy takes the final form of:

$$\rho \frac{Dh}{Dt} = \frac{Dp}{Dt} + div(k\nabla T) + \Phi$$ **[eq 2-3-1]**

where: in addition to the variables defined in equations 2-1 to 2-2 (above),

Φ = dissipation function or deformation heating[12]

k = conduction of continuum

h = enthalpy

p = pressure

The preceding basic equations (eqs 2-1 through 2-3) form the fundamental building blocks for all CFD codes.

2.3.1.2 Boundary conditions: general

In order to solve the flow (***V***, *p*, *T*) of a specified problem, an appropriate set of governing equations and boundary conditions must be selected. It is always prudent to consider solving simplified forms of the Navier-Stokes equations when the simplifications retain the physics, which are essential to the goals of the simulation.

[11] Fourier's law: $q = -k\nabla T$
where: $\boldsymbol{q}$ = the vector heat flow per unit area; k = conduction of the continuum; T = absolute temperature

[12] The dissipation function Φ involves the viscous stresses. It is always positive definite, in accordance with the second law of thermodynamics, since viscosity cannot add energy to the system [9].

Examples of simplified governing equations include steady flows (*d/dt* =0) or incompressible flows (ρ = constant). Boundary types may include solid walls, inflow and outflow boundaries, periodic boundaries, and symmetry boundaries amongst others.

If necessary, physical models must be used for processes, which cannot be simulated within specified constraints (boundary conditions, assumptions). Turbulence is an example of a physical process that is not simulated but rather modelled using empirical information and modifications to the Navier-Stokes equations due to constraints in the Navier-Stokes equations [28].

Furthermore, the success of a simulation depends much on the engineering insight involved in selecting the governing equations, boundary conditions and physical models based on the problem specification. This fact will be elaborated on in the base case formulation in Chapter 4.

2.3.1.3 Discretisation of equations: the CFD approach

Solving the partial differential equations of a flow problem involving complex geometries requires a numerical approach: the complex flow domain needs to be divided into cells or elements. Such a numerical approach requires the tessellation of the flow domain, which is known as a mesh or a grid (in 2 dimensions (2D) or 3 dimensions (3D)). The sum of these cells (in 3D) or areas/elements (in 2D) will equal the flow domain.

Each of these cells can be regarded as a control volume. In order to solve for the flow (for example) in the calculation domain, the differential equations need to be discretised.

A numerical solution of a differential equation consists of a set of numbers from which the distribution of the dependent variables (for example *p*, *T*) can be constructed. This is different from the analytical solution that describes the continuous values of p and T throughout the domain (for example x, y in 2D) – thus an infinite amount of values of the dependent variables.

Discretisation is thus a method that replaces the continuous information contained in the exact solution of the differential equation with discrete values of T and p (following the example) at a finite number of given points in the domain [29]. Employing a suitable discretisation method, the continuum calculation domain can be discretised: the discretisation of space and of the dependent variables makes it possible to replace the governing (partial) differential equations with simple algebraic equations, which can be solved with relative ease.

The discretisation method followed by most CFD codes is the control volume formulation: The calculation domain is divided into a number of non-overlapping control volumes such that there is one finite control volume surrounding one grid point. The differential equations are then integrated over each control volume.

Suppose (as a vehicle for explanation) that there is only one dependent variable ϕ described by a differential equation. Piecewise profiles expressing the variation of ϕ between the grid points are used to evaluate the required integrals: the result is a discretisation equation containing the values of ϕ for a group of grid points. The discretisation equation obtained in this manner expresses the conservation principle for ϕ for the finite control volume, just as the differential equation expresses it for an infinitesimal control volume [29].

The control volume discretisation formulation ensures the integral conservation such as mass, momentum and energy over any group of volumes, and thus over the entire calculation domain.

For more information on developing or deriving control volume discretisation equations for CFD codes, refer to references [28][29][30][31].

2.3.2 Pre-processing: geometry and grid generation

Pre-processing for any CFD flow problem to be solved comprises the preparation of the geometry, as well as dividing of the flow domain or geometry into cells or elements, called the mesh or grid. Pre-processing can thus be summarised as being the process of geometry and (initial)[13] grid generation.

Most commercial CFD packages employ their own pre-processor[14] to generate the geometry and grid. In the event of a design study (or optimisation design), most pre-processors enable users to make use of a parametric grid description, which can be automatically altered by merely adjusting parameters.

Different gridding strategies exist: structured (mostly hexagonal), unstructured (triagonal and pyramids), hybrids, composite and overlapping grids. The choices of numerical methods (discretisation equations) and models (especially turbulence models) to be used, and gridding strategies, are strongly interdependent. The success of a simulation can depend on appropriate choices (gridding strategies and models) for a certain class of problems.

2.3.3 Models in commercial CFD codes

Instead of using the complete set of partial differential equations (based on the complete compressible equations shown in section 2.3.1), "model" equations are used which isolate certain aspects of physics contained in the complete set of equations.

[13] Pre-processing creates an initial grid before the CFD solution process is initiated and started. Most CFD codes include an option to alter the grid based on the solution in progress, also known as solution-adaptive gridding. These grid changes (grid refinement or grid coarsening for example) take place during the solution procedure and are obviously not part of pre-processing in CFD models.

[14] The pre-processor used in this dissertation is GAMBIT [11], which is the pre-processor for FLUENT [10]

In order to save computational time, model equations (simplification assumptions on the flow) can be carefully selected to be representative of the solution of the complete set of equations.

The selection of turbulence models is especially important, as the direct numerical simulation of turbulence is extremely computationally expensive and thus not currently an option. Therefore, different turbulence models are used by CFD codes to "model" the expected influence of turbulence on the flow domain. Accompanying wall functions also need to be defined by the user when a turbulence model is chosen, to assume the boundary layer appearance (as the boundary layer will differ from that of a laminar flow domain).

More detail will be devoted on the choice of turbulence models and other settings in Chapter 4.

In conclusion on CFD simulations (sections 2.3.1 – 2.3.3): the success of the simulation depends almost entirely on engineering insight into the problem:

- intelligent choice of domain boundary conditions and grid strategy is essential;
- choice of CFD turbulence models is essential for representative solution of the class of problem;
- solution-adaptive gridding and convergence selection criteria[15] are essential to ensure physically correct results.

2.3.4 <u>Performance and monitoring criteria (for CFD modelling)</u>

2.3.4.1 Residuals

When evaluating a flow problem using CFD techniques, it is important to constantly monitor the residuals of the solution procedure. The residuals are

[15] More detail will be devoted on the choice of turbulence models and other settings in Chapter 4.

(briefly) the difference between the values of the solution field (velocity, temperature, continuity, and turbulence) for the preceding iteration and the current iteration. Low residuals suggest a solution that converged and can be considered as stable if the residuals keep lowering.

A typical criterion for a converged solution field is residual threshold values required for variables during the iteration process. Mostly, the energy required residual value is below 10^{-6}, and for momentum or continuity at least 10^{-3}. More detail on this matter will be discussed in Chapters to follow, especially Chapter 4.

In most CFD codes, it is possible to accelerate or slow down the changes from iteration to iteration. These methods are called over-relaxation and under-relaxation respectively. Under-relaxation (slowing down changes) is a very useful device for non-linear problems (especially the Navier-Stokes equations). It is often employed to avoid divergence in the iterative solution of strongly non-linear problems [29]. This method will be discussed at a later stage as a very important and useful tool to enforce convergence.

2.3.4.2 <u>Solution monitoring</u>

Low residuals are not a guarantee that the solution is correct. The solution might converge to an incorrect on non-physical[16] flow field answer. It is therefore implorable that other performance criteria are monitored to ensure answers that, in the event of sufficient residual convergence, can be accepted and trusted as physically correct.

It is therefore customary and recommended by CFD coders that a physical quality or variable of the solution flow field is monitored to ensure a true converged solution:

[16] The term non-physical is used to describe a flow field or heat distribution (for example) that is not possible or does not reflect physical reality. The concept of answers that are non-physical is common with CFD analyses and care must be taken to identify when a solution is diverging from reality.

For example, the velocity magnitude on the meniscus-surface (at a specific point) in a model of the continuous casting mould can be monitored. During the initial stages of the numerical computation, the residuals and the variables (physical qualities) will vary with each iteration. If the values of the residuals are sufficiently low, the answer may still not be converged: the velocity magnitude of a certain point on the meniscus surface may still be oscillating or still be asymptotically nearing its final value, indicating a solution that is not sufficiently converged. If the velocity magnitude remained constant for a sufficient[17] number of iterations, and the residuals reached the pre-determined criteria, the answer (flow field) can be assumed to be converged and to consequently represent physical reality.

It is emphasised that a physical property must be monitored to ensure true solution convergence, especially since excessive under-relaxation (by the CFD user) can easily reduce residuals to unrealistically low values without true solution convergence.

2.4 Design Optimisation

The following section offers a general background on design optimisation to the reader who is unfamiliar with this process. More detail information on the specific mathematical optimisation technique used in this dissertation will be presented in Chapter 5.

Design optimisation using CFD modelling encompasses the following processes (in chronological order):

- Base case evaluation

[17] This term is also quite commonly used with CFD calculations, as the sufficient criteria for convergence depends on the type of problem, type of assumptions, type of equations used, type of grid and solution convergence strategy, *inter alia*. In this dissertation, the term "sufficient" will be defined properly in the text whenever used.

- CFD model perfection
- Parameter / variable identification
- Objective function(s) and constraint function(s) identification
- Parameterisation of geometry to be designed
- Optimisation begins:
    - Evaluate perturbations around base case
    - Optimiser[18] predicts new optimum set of points (one design iteration)
    - New perturbations are chosen (by the Optimiser) around optimum set of points – new optimum is predicted after all perturbations (of variables) are evaluated
    - Optimisation continues until objective function is minimised and constraint functions are satisfied sufficiently (more design iterations)
- Experimental evaluation of optimum or final design (if necessary)
- Evaluation of off-design performance of optimum design – robustness of optimum (as in the case with manufacturing tolerances, for example)
- Trade-off studies with regards to certain parameters and variables to obtain true optimum

2.4.1 Base case evaluation and model perfection

Most design optimisation problems involve an existing physical process to be optimised. This existing process is called the base case in the optimisation design process. The logical first step is to evaluate the base case in the CFD code and to compare the results with the real physical process. All the relevant gridding strategies, assumptions made and models chosen in the CFD code can be experimented with to perfect (or to at least closely resemble the physical process) the CFD model of the base case.

[18] The Optimiser refers to a software package (based on a mathematical optimising algorithm) utilised to predict the combination of variables that will minimise the chosen objective function, using the base case information and a set of perturbed base case designs (set of parameters). The Optimiser used in this dissertation is LS-OPT [12]

Evaluating the base case usually gives the user insight into the problem and can lead the user to identify suitable parameters or variables that have a marked influence on the solution flow field.

2.4.2 Parameter and objective and constraint functions identification

The goal of optimisation is to alter certain design variables (selected from process parameters) of an existing process (the base case) in such a way that the "best" combination of parameters (thus design) is found. The "best" design must be quantified: this is achieved by choosing (or developing) an objective function (a function of the parameters or variables). Usually, the objective function is chosen in such a way that the latter must be minimised for the best or optimum solution. Simultaneously, constraint functions are chosen for obvious constraints (e.g., minimum temperature cannot be lower than the solidus temperature) and other design-related constraints as a certain maximum SEN nozzle angle, for example. These constraint functions must be satisfied while minimising the objective function.

For example, if the maximum turbulent kinetic energy on the meniscus surface of a mould in the continuous casting process is to be minimised while limiting the minimum meniscus temperature to prevent freezing, the combination of variables (e.g., nozzle angle with horizontal, submergence depth of nozzle, nozzle port height) that causes the lowest turbulent kinetic energy, is the optimum (and constrained) design.

Further examples of constraint functions are typical bounds (minimum and maximum) for all design parameters, along with other physical constraints as manufacturing tolerances, for example. Monitored quantities such as velocities, temperatures and pressures, or integrals of them on surfaces or in volumes, may also be used as constraints to be satisfied during optimisation.

2.4.3 Parameterisation of CFD model

The more variables to be optimised, the more perturbations are necessary for the optimiser to predict the next optimum design in each design iteration. In this dissertation the amount of variables (and thus number of CFD evaluations needed) forced the author to make use of the scripting capability of the pre-processor GAMBIT [11]: the geometry and mesh generation were parameterised. The Optimiser can now specify a set of variables for a new and unique geometry to be generated by GAMBIT with the parameters as the only input.

Linking the Optimiser, CFD code and pre-processor, the design optimisation process can be started.

2.4.4 Design optimisation [general description]

The first design iteration comprises the evaluation of the base case and the perturbations (of designs) "around" the base case. The objective function value is now known for all these cases. The Optimiser fits a curve or rather hyper surface[19] or approximation through the points (values of objective function as a function of the design variables), and predicts the new combination of variables (thus a design) where the lowest objective function value occurs (according to the curve or approximation fitted through the known points).

For the second design iteration, the optimum achieved in the preceding iteration serves as the new "base case". The same procedure is followed to obtain an optimum design for the second design iteration.

[19] This is a very general description and only applicable to one and two variable optimisation problems: obviously, if there are more than two variables, this "curve" cannot be visualised (in which case it will be a hyper surface).

This process is continued until the objective function converges to a constrained minimum, corresponding to the optimum combination of design variables, also known as the optimum design.

2.4.5 Experimental validation

The optimum design is usually validated by the physical process. For example, if a new design for a Submerged Entry Nozzle (SEN) is suggested, the SEN will typically be built and the CFD results can be validated if compared to the measurements in the real physical continuous casting process.

In the event of a process that will have immense environmental and/or financial implications (as a nuclear reactor design change), other experimental validations of the CFD models can be considered. In this dissertation, the CFD models are validated and compared with full scale and 40%-scaled water models of the continuous casting mould. The validation of the CFD models with water modelling will be discussed in detail in the appropriate Chapter.

2.5 Conclusion of Literature Survey

The brief history of continuous casting of steel was firstly presented to show the progress of this process over the years. Initially, the SEN was not focused on at all due to the many other technical problems that had to be eliminated to enable continuous casting. In an ongoing historical effort to reduce the height of continuous casting machines (consequently lowering the ferrostatic forces and therefore plant costs), a horizontal casting machine is acquired in the limit. However, with the latter set-up, a SEN is substituted for a horizontal refractory nozzle, and therefore falls beyond the scope of this dissertation.

More recently, as indicated by the vertical continuous casting literature, the SEN is recognised as the last component in the continuous casting process, which may have a marked influence on the ultimate quality of the steel. It is therefore an attractive subject for design optimisation.

The continuous casting literature consulted was classified in different categories, namely SEN/mould, tundish, inclusions and ladle literature. The SEN/mould literature (as well as the tundish literature) can be subdivided into the following categories: water modelling, numerical modelling and plant trials. This dissertation will mostly be involved with water modelling and numerical modelling.

Necessary background on numerical modelling and CFD modelling illustrated the basic principles of using computers to model real engineering flow problems. Furthermore, the importance of engineering insight into any CFD modelling exercise was highlighted.

Lastly, a very brief description of general mathematical optimisation was presented as general background to the reader unfamiliar with optimisation techniques.

This concludes the literature survey and should place the dissertation topic (CFD model generation and validation, in an effort to obtain an optimum SEN design using mathematical optimisation techniques) into perspective for the reader.