Modelization of the Melt Flow and the Meniscus Shape in an Induction Furnace and Test of the Velocity Profile with the UDV Probe

Supervisors
Prof. Egbert Baake
Prof. Michele Forzan

Tutors
Thomas Steinberg
Cristian Pozza

Student
Gianmarco Nardo

Academic year 2014/2015
To my family, Benedetta and my friends,
to Padova, Rostock, Hannover.
# Contents

1 Introduction  

2 Physical description  
   2.1 Governing equations  
      2.1.1 The Maxwell's equation  
      2.1.2 Lorentz Force and its application  
      2.1.3 The magnetic vector potential  
   2.2 Fluid Flow analysis  
      2.2.1 Two-equation models  
      2.2.2 The $k$-$\epsilon$ model  
      2.2.3 The $k$-$\omega$ model  
      2.2.4 Boundary conditions  
      2.2.5 The SST $k$-$\omega$ model  
      2.2.6 The Low Reynolds number $k$-$\epsilon$ model  

3 Numerical Simulation  
   3.1 Description of the furnace  
   3.2 Sets of the computation  
   3.3 The meniscus shape  
   3.4 Fluid Flow modelization  
      3.4.1 the $k$-$\epsilon$ and the $k$-$\omega$ solution  
      3.4.2 the $k$-$\epsilon$ solution  
      3.4.3 the $k$-$\omega$ solution  
      3.4.4 The Low-Re $k$-$\epsilon$ solution  

4 Measurements  
   4.1 The pulsed Ultrasound Doppler Velocimetry  
   4.2 The Ultrasonic Field  
   4.3 Comparison of measurements and computed velocities  

5 Conclusions
CONTENTS

A Integrative Formulas 75
   A.1 Integration function 75
   A.2 Curvilinear description of the Lorentz Force 75
List of Figures

2.1 Oscillating and Mean velocity, Comsol© ........................................... 7
2.2 Simple description of the velocity in presence of walls ............................ 9
2.3 The four regimes of turbulent flow, Comsol© ...................................... 11

3.1 The modeled geometry ........................................................................ 15
3.2 Diagram of the study ........................................................................... 15
3.3 description for the equation (3.1) .......................................................... 16
3.4 Point distribution .................................................................................. 17
3.5 Block structure of the project .................................................................. 18
3.6 Height deformation after iterations .......................................................... 18
3.7 Convergence of the pressure [atm] .......................................................... 19
3.8 Mesh deformation with 10000 [A] ............................................................ 19
3.9 Mesh deformation with 10000 [A] with finer mesh ................................... 20
3.10 computed Lorentz Force with deformed and undeformed domain .......... 21
3.11 Turbulence description near the free surface for the coarse bounded and a
fine mesh ..................................................................................................... 22
3.12 Eddy viscosity ratio for k-ԑ RANS solution with a bounded mesh .......... 23
3.13 Eddy viscosity ratio for k-ԑ RANS solution with a full mapped mesh .... 25
3.14 k-ԑ Vortexes with velocity profile the deformed mesh ...................... 27
3.15 k-ԑ Vortexes with velocity profile without the deformed mesh .......... 28
3.16 k-ԑ turbulent kinetic energy without the deformed mesh .................. 29
3.17 k-ԑ turbulent kinetic energy with the deformed mesh ...................... 30
3.18 Eddy viscosity ratio without the deformed mesh .................................. 31
3.19 Comparison of axial velocities for the k-ԑ solution at different radius ... 33
3.20 k-ω Vortexes with velocity profile without the deformed mesh .......... 35
3.21 k-ω Vortexes with velocity profile with the deformed mesh .............. 36
3.22 k-ω turbulent kinetic energy without the deformed mesh .................. 37
3.23 k-ω turbulent kinetic energy with the deformed mesh ...................... 38
3.24 k-ω eddy viscosity ratio without the deformed mesh ......................... 39
3.25 k-ω eddy viscosity ratio with the deformed mesh ............................... 40
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.26</td>
<td>Comparison of axial velocities for the $k$-$\omega$ solution at different radius</td>
<td>42</td>
</tr>
<tr>
<td>3.27</td>
<td>Low Reynolds $k$-$\epsilon$ Vortices with velocity profile without the deformed mesh</td>
<td>44</td>
</tr>
<tr>
<td>3.28</td>
<td>Low Reynolds $k$-$\epsilon$ Vortices with velocity profile with the deformed mesh</td>
<td>45</td>
</tr>
<tr>
<td>3.29</td>
<td>The kinetic Low Reynolds $k$-$\epsilon$ energy without the deformed mesh</td>
<td>46</td>
</tr>
<tr>
<td>3.30</td>
<td>The kinetic Low Reynolds $k$-$\epsilon$ energy with the deformed mesh</td>
<td>47</td>
</tr>
<tr>
<td>3.31</td>
<td>The Low Reynolds $k$-$\epsilon$ eddy viscosity ratio without the deformed mesh</td>
<td>48</td>
</tr>
<tr>
<td>3.32</td>
<td>The Low Reynolds $k$-$\epsilon$ eddy viscosity ratio with the deformed mesh</td>
<td>49</td>
</tr>
<tr>
<td>3.33</td>
<td>Comparison of axial velocities for the Low Reynolds $k$-$\epsilon$ solution at different radius</td>
<td>51</td>
</tr>
<tr>
<td>4.1</td>
<td>Description of the acquisition ([19])</td>
<td>55</td>
</tr>
<tr>
<td>4.2</td>
<td>The Ultrasonic Field ([19])</td>
<td>56</td>
</tr>
<tr>
<td>4.3</td>
<td>Oscillation related to the Field Regions</td>
<td>57</td>
</tr>
<tr>
<td>4.4</td>
<td>Error in measured frequency ([19])</td>
<td>58</td>
</tr>
<tr>
<td>4.5</td>
<td>Comparison of axial velocities for the $k$-$\omega$ solution at different radius</td>
<td>67</td>
</tr>
<tr>
<td>4.6</td>
<td>Comparison of axial velocities for the Low Reynolds $k$-$\epsilon$ solution at different radius</td>
<td>71</td>
</tr>
</tbody>
</table>
1. Introduction

The Magnetofluidodynamics (MFD) field examines the dynamics of electrically conducting fluids. It is a science that began its development with Hannes Alfvén (Nobel Prize in 1970) and from which various disciplines grew as well as applications, for example for particle accelerators, controlled thermonuclear fusion and for astrophysics concepts. From an industrial engineering point of view, more attention has been paid to the energy production sector, for example the MHD generator, or to the heating of conductive materials. The last one shows correlations with classical electrothermics, that usually does not not consider the study of the electric motion. The most known case is the induction furnace where, the magnetic fields created by an external inductor, generate heat thanks to the joule effect produced by the induced currents. Depending on the metal or the kind of alloys to melt, various types of furnaces are adopted. Several parameters must be chosen to properly design an induction furnace, from the the choice of the electrical feed to the materials that constitute a furnace. Since creating experimental or scaled furnace is expensive, computer simulation helps on predicting the most correct geometry and overall parameters. Computational Fluid Dynamics finds application on several problems and covers different branches of physics, such as aerodynamics, fluid flow in pipes and biomedical problems. Because of this versatility, various types of equations are given for the same problem, that may not give the equal result. The approach called RANS, or the Reynolds Averaged Navier Stokes, is probably the most applied for solving industrial problems, because it has inexpensive computational cost but it suffers from lower generality and models must be defined differently according to the problem. In this case the turbulent flow is seen as a contribution to a time-averaged quantity and the fluctuations over this medium value: with the use of transport equations and assumptions made for Reynolds stress tensor, the correlation between these two terms is obtained, reducing the computational cost because the scales of the time-averaged variable are bigger than of the turbulent flow. The Institute of Electrotechnology of Hanover actually works on the development of the LES technique that requests much more computational power than a RANS procedure but allows more precise solution, because LES makes reference to universal models that have a bigger prediction capacity. It consists on calculating numerically the behavior of the biggest scales of the flow and filter the smallest scales
directly dependant on those, using ad-hoc subgrid scale models to establish their effect. This work is focused on finding the most correct modeling for a typical induction furnace, in this thesis the furnace installed at the institute ETP is taken as reference, and the calculated velocity distributions are compared with measurements made by an Ultrasonic Doppler Velocimetry probe. The Wood’s metal was the molten fluid adopted for this study because of its low-melting point. The computer simulation was done with Comsol© that works exclusively with RANS equations and provides different types of transport equations, such as $k$-$\epsilon$, $k$-$\omega$, Low Reynolds $k$-$\epsilon$, SST $k$-$\omega$ and Spalart-Allmaras.

- Chapter 2 describes the equations that hold up the electromagnetic field and the fluid flow dynamic problem, with interests to the boundary fluid flow condition and the effects of the Lorentz Force.

- Chapter 3 explains the numerical model built for the IF problem. It is subdivided in the analysis of the computation of the free surface shape, and the comparison of the RANS $k$-$\epsilon$, $k$-$\omega$, Low Reynolds $k$-$\epsilon$ solution. Furthermore, models have been developed taking into account the presence of the meniscus as well as considering the upper free surface as unmodified.

- Chapter 4 gives a description of the UDV instrument and shows comparisons between the acquired velocity measures with the computed ones.
2. Physical description

This chapter describes the equations that hold up the electromagnetic and the fluid dynamic problems, with interests to the boundary fluid flow condition and the meaning of the Lorentz Force.

2.1 Governing equations

2.1.1 The Maxwell’s equation

The MHD formulation starts considering the electromagnetic phenomenons, described by the Maxwell’s equations. For materials which are neither magnetic nor dielectric, and assuming that the liquid metal has a large electrical conductivity, the displacements currents can be ignored as well as the accumulation of the electrical charge on the medium. As a consequence, the electric field induced by the motion of the charges is negligible. The simplified Maxwell’s equations in differential form so state:

\( \nabla \times H = J \)  \hspace{1cm} (Ampere’s Law) \hspace{1cm} (2.1)

\( \nabla \cdot J = 0 \)  \hspace{1cm} (Charge Conservation) \hspace{1cm} (2.2)

\( \nabla \cdot B = 0 \)  \hspace{1cm} (Solenoidal nature of B) \hspace{1cm} (2.3)

\( \nabla \times E = -\frac{\partial B}{\partial t} \)  \hspace{1cm} (Faraday’s Law) \hspace{1cm} (2.4)

Maxwell’s equations provide a number of unknowns that run over the number of equations. The properties of a medium show correspondences between them, and with the same hypothesis described before, the lead constitutive relations are as follows:

\( B = \mu_0 (H) \)  \hspace{1cm} (2.5)

\( J = \sigma (E) \)  \hspace{1cm} (2.6)
To complete the information it’s also necessary to define the Lorentz force contribution:

\[ F = J \times B \] (2.7)

Combining Ohm’s Law (2.6), the Faraday’s (2.4) and the Ampere’s Law (2.1), it’s possible to gain an expression relating \( B \) to \( u \):

\[ \frac{\partial B}{\partial t} = -\nabla \times E = -\nabla \times \left[ \left( \frac{J}{\sigma} \right) - u \times B \right] = \nabla \times \left[ u \times B - \nabla \times \frac{B}{\mu \sigma} \right] \]

As the magnetic field \( B \) is solenoidal (2.3), this simplifies to:

\[ \frac{\partial B}{\partial t} = \nabla \times \left( u \times B \right) + \lambda \nabla^2 B \] (2.8)

This equation is called the advection-diffusion equation for \( B \) and the quantity \( \lambda \) is noted as the magnetic diffusivity \([m^2/s]\): in the case \( u \) is defined, then \( B \) is spatially and temporally predicted from initial conditions. In our case, it is customary to introduce the dimensionless number called the Magnetic Reynolds number, \( R_m \), directly comparable to the normal Reynolds number, to evaluate the importance of the advection term and diffusion. Indeed, the normal Reynolds number is considered as the magnitude of the vorticity term

\[ \nabla \times (u \times \omega) \sim (U/L) \omega \] (2.9)

that comes from the vorticity evolution equation,

\[ \frac{\partial \omega}{\partial t} = \nabla \times (u \times \omega) + \nu \nabla^2 \omega \] (2.10)

made in comparison with the magnitude of the diffusion term \( \nu \nabla^2 \omega \sim (\nu/L^2) \omega \):

\[ R = (U/L) \left( \frac{\nu}{L^2} \right)^{-1} = UL\nu \] (2.11)

where \( V \) is the characteristic length speed, \( L \) is the characteristic lengthscale of the flow and \( \nu \) the viscosity. \( R_m \) is the ratio of the the magnetic-flux-freezing term \( \nabla \times (u \times B) \sim (U/L) B \) to the magnitude of the magnetic diffusion term \( D_m \nabla^2 B = (1/\mu_0 \sigma e L^2) \) in the induction equation (2.8):

\[ R_m = \frac{U/L}{\lambda/L^2} = \frac{UL}{\lambda} = \mu_0 \sigma e UL. \] (2.12)

In a situation where \( R_m \gg 1 \), the field lines are regarded as frozen with the fluid [3], while our case leads to \( R_m \ll 1 \) when the ohmic current is dominant: the resistivity is therefore significant and so the diffusivity is rarely negligible, furthermore the flow is
driven by the magnetic field but does not influence the magnetic field significantly in return. When an oscillating electromagnetic field interests a medium with conductivity $\sigma$ and permeability $\mu_r$, induced oscillating currents are established and spread according to the depth of penetration:

$$\delta = \sqrt{\frac{2\rho}{\omega \mu_0 \mu_r}}$$

(2.13)

The surface current shields the external magnetic field by inducing a new magnetic field which is in the opposite direction of the external field. Interference between the two magnetic fields results in the exponential decay of the external magnetic field.

### 2.1.2 Lorentz Force and its application

In the case of linear materials, Lorentz forces could be described in their rotational and irrotational components. Using the vector identity:

$$\nabla \frac{B^2}{2} = (B \cdot \nabla) B + B \times \nabla \times B$$

we define the equation (2.1) as:

$$\mathbf{J} \times \mathbf{B} = (B \cdot \nabla) \left( \frac{B}{\mu} \right) - \nabla \left( \frac{B^2}{2\mu} \right)$$

(2.14)

The first term on the right side of the equation interests the movement of the fluid, it can be eventually decomposed in two ways depending on the purpose of the study [6]. The second term $-\nabla (B^2/2\mu)$ is an irrotational component that acts as a lifting force and it concerns the gradient of the magnetic pressure: it gets no influences on the flow field and gives no contributions to the vorticity equation. Considering a hydrostatic steady state, the condition of the equilibrium free surface of a molten metal will be as follows:

$$\rho gh_t = \frac{B^2}{4\mu} + K\gamma + C$$

with $\gamma$ as the surface tension, $C$ an integration constant usually kept to zero, $h_t$ is measured from the top height of the molten metal to the point under consideration, $B$ is the mean square value of the magnetic field evaluated at top of the molten metal and $K$ the sum of the curvature of the free surface. Because of the frequency and the current applied to the model ([13], [23]), it is usual to ignore the surface tension and and consider only the hydrostatic and the magnetic pressure. At equilibrium, it must be that:

$$\rho gh_t = \frac{B^2}{2\mu}$$

(2.15)
2.1.3 The magnetic vector potential

Magnetic fields obtained by steady currents satisfy (2.4) and allows to write:

\[ \mathbf{B} = \nabla \times \mathbf{A} \]  

(2.16)

hence the divergence of a curl is automatically zero. As a consequence, substituting (2.2) and (2.5) into (2.1), involves the first order differential equation:

\[ \nabla \times \left( \frac{1}{\mu} (\nabla \times \mathbf{A}) \right) = \mathbf{J} \]  

(2.17)

This is a general equation that describes the electromagnetic fields in term of \( \mathbf{A} \) which \( \mathbf{J} \) is the current density including both the impressed and the induced components. Any function \( \mathbf{A}' = \mathbf{A} + \nabla f \) yields the solution of (2.17), therefore to define it uniquely, a gauge condition has to be superimposed:

\[ \nabla \cdot \mathbf{A} = 0 \]

The combination of (2.4) and (2.6), with the introduction of the scalar potential \( V \), leads the equation:

\[ \mathbf{J} = -\sigma \frac{\partial \mathbf{A}}{\partial t} - \sigma \nabla V \]  

(2.18)

Regarding to the right size, the first term concerns the induced current density and the second term is the impressed current density, and so (2.17) may be written as:

\[ \nabla \times \nabla \times \mathbf{A} + \mu \sigma \frac{\partial \mathbf{A}}{\partial t} = \mu \mathbf{J}_i \]  

(2.19)

with \( \mathbf{J}_i \) as the only induced current. Working with \( \mathbf{A} \) needs at the bounday of the whole geometry a magnetic insulation connotation:

\[ n \times \mathbf{A} = 0 \]  

(2.20)

2.2 Fluid Flow analysis

In the most metallurgical applications as the melting furnaces, the value of the Reynolds number exceeds its critical value \( \text{Re}_{\text{critical}} \). Because of this consequence, the study must take care of a turbulent model instead of a laminar flow, applying the Reynolds-Averaged Navier Stokes equation. Derived from the classical Navier Stokes equation, the variables are represented in their time-averaged variable \( \overline{X} \) and turbulence component
2.2. FLUID FLOW ANALYSIS

\[ U = \bar{U} + U' \]

\[ \bar{U} = \lim_{T \to \infty} \left( \frac{1}{T} \int_0^{0+T} U \, dt \right) \]

Figure 2.1: Oscillating and Mean velocity, Comsol©

Consequently the equations, holding the conservation of mass and momentum, taking into account an incompressible flow for an unsteady solution, are:

\[ \nabla \cdot U = 0 \]  \hspace{1cm} (2.21)

\[ \rho \frac{\partial U}{\partial t} + \rho \nabla (UU) + \rho \nabla \left( u' u' \right) = -\frac{\partial P}{\partial x} + \mu \nabla^2 U \]  \hspace{1cm} (2.22)

and, for \( i, j \) unit vectors of a 3-dimensional flow \( u, v, w \), it results as follows:

\[ \frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial u_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( -\rho \delta_{ij} + \mu \frac{\partial \bar{u}_i}{\partial x_j} - \rho u'_j u'_i \right) \]  \hspace{1cm} (2.23)

With the introduction of the eddy viscosity, the last term describing the Reynolds stresses could be represented with the Boussinesq assumption:

\[ \rho u'_i v'_j = -\mu_t \left( \frac{\partial \bar{V}_i}{\partial U_j} + \frac{\partial \bar{V}_j}{\partial x_i} \right) + \frac{2}{3} \rho k \delta_{ij} \]  \hspace{1cm} (2.24)

with \( k \) defined as the specific turbulent kinetic energy:

\[ k = \frac{1}{2} \bar{u}_i u_j = \frac{1}{2} \left( \bar{u}^2 + \bar{v}^2 + \bar{w}^2 \right) \]  \hspace{1cm} (2.25)

and \( \delta_{ij} \) as the Kronecker delta (\( \delta_{ij} = 1 \) if \( j = i \), 0 elsewhere). This last term is the Kolmogorov assumption for local isotropy that serves as a description of the dissipation at sufficient small turbulence length. With the turbulent viscosity the number of variables occurring for calculating the Reynolds stresses decades from six for each component of the Reynolds-stress tensor to one.
2.2.1 Two-equation models

Resolving (2.22) needs aim from algorithms that have been modeled and defined in the last seventy years. The most commonly used for industrial application are the two-equation models, that adds two transport equations and two more variables.

2.2.2 The $k$-$\epsilon$ model

This model uses the turbulent dissipation rate $\epsilon$ and the kinetic energy in order to gain the turbulent viscosity throughout the equation:

$$\mu_t = \rho C_{\mu} \frac{k^2}{\epsilon} \tag{2.26}$$

The quantity $\epsilon$ refers to the dissipation per unit mass and is defined by the correlation:

$$\epsilon = \nu \frac{\partial u_i}{\partial x_j} \frac{\partial u_i}{\partial x_j} \tag{2.27}$$

Since working with fluctuations is complex, and moreover because it's requested to predict properties for a turbulent flow with no prior knowledge of the turbulent structure, two transport equations for $\epsilon$ and $k$ support the computation:

$$\frac{\rho \partial k}{\partial t} + \rho \nabla (k U) = \nabla \left( \mu_t \nabla k \right) + 2 \mu_t E : E - \rho \epsilon \tag{2.28}$$

$$\frac{\rho \partial \epsilon}{\partial t} + \rho \nabla (k U) = \nabla \left( \frac{\mu_t}{\sigma_{\epsilon}} \nabla \epsilon \right) + C_{1\epsilon} \frac{\epsilon}{k} 2 \mu_t E : E - C_{2\epsilon} \rho \frac{\epsilon^2}{k} \tag{2.29}$$

These equations help finding the eddy viscosity and the turbulence length scale \[22\]. The model presents closure constants:

$$C_{\mu} = 0.09, \quad C_{1\epsilon} = 1.44, \quad C_{1\epsilon} = 1.92, \quad \sigma_{\epsilon} = 1.3, \quad \sigma_k = 1.0 \tag{2.30}$$

2.2.3 The $k$-$\omega$ model

The $k$-$\omega$ model solves for $k$ and the specific dissipation rate, $\omega$, that is the dissipation per unit of volume and time of the turbulent kinetic energy. The turbulent viscosity is determined by the following equation:

$$\mu_t = \rho \frac{k}{\omega}$$
2.2. FLUID FLOW ANALYSIS

The two transport models are written below:

\[
\frac{\rho \partial k}{\partial t} + \rho \mathbf{u} \cdot \nabla k = P_k - \rho \beta^* k \omega + \nabla \cdot ((\mu + \sigma^* \mu_t) \nabla k) \tag{2.31}
\]

\[
\frac{\rho \partial \omega}{\partial t} + \rho \mathbf{u} \cdot \nabla \omega = \alpha \frac{\omega}{k} P_k - \rho \beta \omega^2 + \nabla \cdot ((\mu + \sigma \mu_t) \nabla \omega) \tag{2.32}
\]

with the closure coefficients

\[
\alpha = \frac{5}{9}, \quad \beta = \frac{3}{40} \quad \beta^* = \frac{9}{100}, \quad \sigma = \frac{1}{2}, \quad \sigma^* = \frac{1}{2}. \tag{2.33}
\]

2.2.4 Boundary conditions

- **Slip**

With a slip boundary condition, no viscous effect are present and the fluid shall not exit the domain:

\[
\mathbf{u} \cdot \mathbf{n} = 0
\]

Considering the viscous stress:

\[
\mathbf{K} = \left[ (\mu + \mu_t) \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) - \frac{2}{3} \rho k l \right] \mathbf{n} \tag{2.34}
\]

\[
\mathbf{K} - (\mathbf{K} \cdot \mathbf{n}) = 0 \tag{2.35}
\]

these means there are no viscous stress in the tangential direction. \( k, \epsilon \) and \( \omega \) follow normal Neumann condition:

\[
\nabla k \cdot \mathbf{n} = 0 \quad \nabla \epsilon \cdot \mathbf{n} = 0 \quad \nabla \omega \cdot \mathbf{n} = 0
\]

- **Wall function**

Let us consider an infinite plane with a velocity gradient only in the \( y \) direction. The Reynolds number is \( Re_y = y U / \nu \). In the case \( y \) is small, \( Re_y \approx 1 \), the viscous forces

![Figure 2.2: Simple description of the velocity in presence of walls](image)
become relevant and as a consequence, the turbulence dissipation. The total (viscous and turbulent) stress is as follows:

\[ \tau_{yx} = \rho v \frac{dU}{dy} - \rho u' v' \]  \hfill (2.36)

At the wall \( u = 0 \), and so:

\[ \tau_{yx} (0) = \rho v \frac{dU}{dy} \equiv \tau_w \]  \hfill (2.37)

\[ u_r \equiv \sqrt{\frac{\tau_w}{\rho}} \]  \quad (Friction velocity) \hfill (2.38)

Applying the definition of viscous distance \( y_w \equiv v/u_r \):

\[ y^+ = \frac{y u_r}{v} \equiv \frac{y}{y_w} \]  \quad (dimensionless distance) \hfill (2.39)

\[ u^+ = \frac{U}{u_r} \]  \quad (dimensionless velocity) \hfill (2.40)

The so called Wall function correlates \( y^+ \) and \( u^+ \) in the way \( u^+ = f_w(y^+) \). With large Reynolds numbers in proximity of the wall, there’s a viscous sub-layer where the velocity profile depends only on the viscous phenomenons and doesn’t interests the mean flow. Therefore:

\[ \tau_w = \tau_{yx} = \rho v \frac{dU}{dy} \]  \hfill (2.41)

After the viscous sub-layer, the flow in the buffer layer begins to be turbulent \( (5 \leq y^+ \leq 30) \): it is pretty deep to host turbulent viscous dissipation at low frequency. A log-wall between the free stream region and the wall \( (30 \leq y^+ \leq 300) \) is normally computed by numerical programs and correlates the mean velocity value to the log distance of the wall. \( K-\epsilon \) and \( k-\omega \), using a proper mixing length model \([4]\) and so the logarithmic wall function, ignore the existence of the buffer layer and extend the values gained from the log-wall directly to the wall, adapting the logarithmic decrease and gaining a non zero velocity value at the wall:

\[ u^+ = \frac{1}{\kappa} \ln y^+ + B \]  \hfill (2.42)

with the constant of van Karman \( \kappa = 0.41 \) and \( B = 5.5 \).
2.2. FLUID FLOW ANALYSIS

Figure 2.3: The four regimes of turbulent flow, Comsol©

The wall function bridges the resolution of the turbulence model in the buffer layer approaching the wall, hence it starts solving from a region $\delta_ω$ that is correlated to this equation:

$$\delta_ω^+ = \frac{\rho}{\mu} \tau_ω \delta_ω$$

$k$, $\epsilon$ and $\omega$ are computed within the wall functions, by prescribed functions [22]:

$$k = \frac{u^2}{\sqrt{\beta^*}}, \quad \omega = \frac{k^{0.5}}{((\beta^*)^{0.25})}, \quad \epsilon = \frac{k^{1.5}}{\kappa y}$$

(2.43)

2.2.5 The SST $k$-$\omega$ model

A good compromise between the $k$-$\omega$ and the $k$-$\epsilon$ is the SST turbulence model: The use of a $k$-$\omega$ formulation in the inner parts of the boundary layer makes the model directly usable all the way down to the wall through the viscous sub-layer. The SST formulation also switches to a $k$-$\epsilon$ behaviour in the free-stream and thereby avoids the common $k$-$\omega$ problem that the model is too sensitive to the inlet free-stream turbulence properties.

Listed as made before:

$$\mu_t = \frac{\rho a_1 k}{\max(a_1 \omega, SF_2)}$$

(2.44)

$$\rho \frac{\partial k}{\partial t} + \rho u \cdot \nabla k = P - \rho \beta_0^* k \omega + \nabla \cdot ((\mu + \sigma_k \mu_t) \nabla k)$$

(2.45)

$$\rho \frac{\partial \omega}{\partial t} + \rho u \cdot \nabla \omega = \frac{\rho \gamma}{\mu_t} P - \rho \beta \omega^2 + \nabla ((\mu + \sigma_\omega \mu_t) \nabla \omega) + 2(1 - F_1) \frac{\rho \sigma_\omega^2}{\omega} \nabla \omega \cdot \nabla k$$

(2.46)

with $F_1$, $F_2$, $P$ variables and $\sigma_\omega$ a constant coefficient described in Menter [14].
• Wall boundary conditions

SST, as well as other Low-Reynolds models, does not involve approximation at the wall while the equations are calculated all the way through the boundary layer to the wall. Remains the fact that \( u, k \) must be zero at the walls but on regions strictly close to the walls, so \( \omega \) is not directly calculated but its boundary condition:

\[
\lim_{l_w \to 0} \omega = \frac{6\mu}{\rho\beta_1 y^2}
\]

(2.47)

The slip condition is the same as before.

2.2.6 The Low Reynolds number \( k-\epsilon \) model

This model is a refinement of the \( k-\epsilon \), with the use of damping functions for regions close to walls where viscous effects dominates:

\[
\mu_t = \rho f_\mu C_\mu \frac{k^2}{\epsilon}
\]

(2.48)

\[
\rho \frac{\partial k}{\partial t} + \rho u \cdot \nabla k = \nabla \cdot \left( \left( \mu + \frac{\mu t}{\sigma_k} \right) \nabla k \right) + P_k - \rho \epsilon
\]

(2.49)

\[
\rho \frac{\partial \epsilon}{\partial t} + \rho u \cdot \nabla \epsilon = \nabla \cdot \left( \left( \mu + \frac{\mu t}{\sigma_\epsilon} \right) \nabla \epsilon \right) + C_{\epsilon 1} \frac{\epsilon}{k} P_k - f_\epsilon C_{\epsilon 2} \frac{\epsilon^2}{k}
\]

(2.50)

with the closure coefficients

\[
C_{\epsilon 1} = 1.5, \quad C_{\epsilon 2} = 1.9 \quad C_{\epsilon 2} = 9/100, \quad \sigma_k = 1.4, \quad \sigma_\epsilon = 1.4
\]

(2.51)

and \( P_k, f_\mu, f_\epsilon \) written in the AKN model [2].

• Wall boundary conditions

The use of damping terms refers especially to \( \epsilon \), since the \( k \), related to \( u = 0 \) on the wall, is either 0. The boundary condition for \( \epsilon \) uses a first approximation of the correct wall boundary condition [5], since its analysis is very unstable:

\[
\epsilon = \frac{2\mu k}{\rho y^2}
\]

(2.52)

There are no changes of concepts for the slip boundary condition.
3. Numerical Simulation

This chapter explains the numerical model built for the IF problem. After introducing the structure of the furnace used for it, the assumptions applied for the simulation are explained and then a view over the different two equation models solution.

3.1 Description of the furnace

The furnace installed at Institute of Electrotechnology of Hanover and developed by professor Baake is suitable for didactic studies over the fluid flow of molten metal. It consists of a pre-furnace, where the metal is heated and fused, and then with the communicating vessels principle the fluid is injected in the furnace from a hole placed at the bottom. The furnace itself has a inner radius of 158 mm and a height of 756 mm and the total inductor height is about 570 mm. The walls of the furnace are made of alloy and the solenoidal rods (11 bars), that surround them, of copper.

These are connected to a current-controlled generator, that was set to $I_{eff} = 2000 \,[A]$ and $f = 385 \,[Hz]$.

<table>
<thead>
<tr>
<th>Property</th>
<th>Symbol</th>
<th>Units</th>
<th>Alloy</th>
<th>Copper</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>$\rho$</td>
<td>$kg/m^3$</td>
<td>7850</td>
<td>8700</td>
</tr>
<tr>
<td>Electrical conductivity</td>
<td>$\sigma$</td>
<td>S/m</td>
<td>$1.4*10^6$</td>
<td>$6*10^7$</td>
</tr>
<tr>
<td>Relative permeability</td>
<td>$\mu_r$</td>
<td>-</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Relative permittivity</td>
<td>$\varepsilon_r$</td>
<td>-</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Thermal conductivity</td>
<td>$k_t$</td>
<td>W/(m·K)</td>
<td>44.5</td>
<td>400</td>
</tr>
<tr>
<td>Heat capacity</td>
<td>$C_p$</td>
<td>J/(kg·K)</td>
<td>475</td>
<td>385</td>
</tr>
</tbody>
</table>

The Wood’s metal was chosen for this study, considering basically its low temperature melting point (above $82^\circ$).

Its material characteristics are described below (\[9\] [3]):
CHAPTER 3. NUMERICAL SIMULATION

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Units</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>ρ</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Electrical conductivity</td>
<td>σ</td>
<td>S/m</td>
</tr>
<tr>
<td>Relative permeability</td>
<td>$μ_r$</td>
<td>-</td>
</tr>
<tr>
<td>Relative permittivity</td>
<td>$μ_r$</td>
<td>-</td>
</tr>
<tr>
<td>Kinematic viscosity</td>
<td>$μ$</td>
<td>Pa·s</td>
</tr>
<tr>
<td>Surface tension</td>
<td>$γ$</td>
<td>N/m</td>
</tr>
</tbody>
</table>

3.2 Sets of the computation

Comsol® 4.4 has been used for solving the electromagnetic and the fluid dynamic models and their interaction, a finite element methods that solves the PDE problems that govern the two physics.

The study required several assumptions to simplify at first the problem:

- the electromagnetic field is steady-state and harmonic
- the coil helicity is ignored
- the temperature distribution of the molten metal is uniform
- the fluid is newtonian, incompressible and always liquid. No changes to solid state are evaluated, therefore the material properties are kept constant
- Buoyancy effects of temperature gradients are neglected due to their low value in respect to the electromagnetic forces
- the Marangoni effect on thermocapillary convection (gained from temperature dependence of the surface tension) is ignored
- the geometry of the model is two dimensional and axysimmetrical
- the motion of the fluid does not interfere with the magnetic field, in the way that $u$ and $B$ are indipendent from each other
- the roughness of the wall is ignored

The computed geometry is showed on the next page. Around the furnace a semicircular element was created with a radius of 1.2 [m], in order to close the magnetic computation in a reasonable area.

Waiting for a moment to discuss about the fluid domain, the mesh adopted for the rest of the geometry was computed by the simulator program Comsol® with a fine mesh instance, except for the copper bar, where a fine mapped mesh was set to compute with the
minimum initial error the magnetic field, and for the molten metal, where a refinement on the region of the depth of penetration was important for giving sufficient points to calculate the inducted current. By the study of the decomposition of the Lorentz Force the irrotational term is not involved the RANS equation, as it deforms only the free surface and doesn’t add sufficient gradient divergence to having significant changes. Another aspect is that Comsol\textsuperscript{©} doesn’t manage to compute the first term of the cited equation, therefore it was necessary for the fluid flow analysis to study $\mathbf{J} \times \mathbf{B}$ without decomposition and the meniscus shape only with the irrotational term. The two physics were studied separated, and then integrated in the following way:

![Diagram of the study](image)

### 3.3 The meniscus shape

The Arbitrary Lagrangian-Eulerian Formulation (ALE) was applied for the formulation of the deformation of the free surface of the molten metal. Following the concept that the
partial differential equations of physics could be formulated either in a spatial coordinate system, with coordinates fixed in space (Eulerian), or in a material coordinate system where the viewer follows the infinitesimal element of the fluid as it deforms (Lagrangian), the first approach shows more robustness for mechanical problems or for fluid that are bounded in containers and when displacements are not so big, meanwhile the second is more suitable for domains without free surfaces or moving boundaries [17]. The coupling of this approach creates a moving mesh grid that follows the fluid domain with a velocity $v$, equal to neither zero nor the velocity of the fluid particles and it changes arbitrarily between both of them [17].

The Lagrangian description works in mesh elements where small motion happens, and the Eulerian description in zones and directions where it’s impossible for the mesh to follow the motion of the fluid. The displacement of the molten metal consists on applying to the free surface the following identity:

$$dz = z_0 - z + h_t$$ \hspace{1cm} (3.1)

where $z_0$ is the fluid top height, $z$ is the previous height, $h_t$ the height difference gained from equation (2.15) and so described:

$$h_t = -\frac{B^2}{2\rho g \mu}$$ \hspace{1cm} (3.2)

![Figure 3.3: description for the equation (3.1)]
3.3. THE MENISCUS SHAPE

$B$ is obtained form a time harmonic solution and is the root mean square value. Its value is sent to the deformed mesh physics and the solver starts computing the mesh displacement using an iteration solver. $z_0$ is calculated within the ordinary differential equation module (ODE) and guarantees the mass conservation. A set of points are distributed along the free surface, and called from the ODE with their vertical component information as a weight for the integration formula.

![Figure 3.4: Point distribution](image)

Considering simply the area of the domain, that shall be equal before and after the solver processing:

$$\int f(x)dx = \text{Area}_i$$

with $f(x)$ the function positioned on the free surface and described in Appendix [A.1].

A direct solver (PARDISO) computes iteratively in a stationary solver the magnetic field and the displacements using every time the new deformed geometry obtained from the precedent step, until the equation (3.2) is satisfied. Previous attempts with Comsol© for a hydrostatic solution [13] faced the problem considering the electromagnetic problem and the deformed mesh problem using the same geometry model. That required a continuity equation on the moving mesh module for spreading the deformation along the wall with boundary mesh displacements that affect the overall solution because the magnetic field, especially in the connection point between the wall and the fluid, could lead to discontinuities whenever the mesh grid is kept after the iterations and that a smoothing (ppr) operator should repair.

The idea was to study the magnetic field in the whole geometry and then, with a general extrusion application, transpose the magnetic pressure information to a model which contains only the studied domain: a specific solver was adopted, where the deformed mesh after each iteration of it was bypassed to the complete model until (3.2) has not been satisfied. As a consequence, punctual material information were re initialized every time.
CHAPTER 3. NUMERICAL SIMULATION

Figure 3.5: Block structure of the project

This helps also to choose for a quadratic geometric shape order, not linear, to reproduce for every element a reasonable shape.

The relative tolerance was set to $10^{-3}$: lower values could not be reached by the solver with non linear geometric shape order. The mesh adopted was mapped, identical between the two models and finer as much as possible, in order to limit the discontinuity of the magnetic field and give more information to the free surface shape: the averaged value is estimated (around $3.21 \times 10^{-6}$ m$^2$):

Figure 3.6: Height deformation after iterations
When using a linear geometric shape, every element cannot adapt itself accurately like with a higher order, hence this corresponds to local errors of the whole deformation. This formulation remains suitable when the amplitude of the magnetic field is not so huge. In such cases the current of the coil is higher than 10 kA, inverted mesh are generated even with a non correlation to the quality of the mesh.
Comparison with higher currents demonstrates that the direct method for the nonlinear solver used for our main study is not suitable. Iterative solvers (bz. GMRES) and transient studies manage to decrease the divergence of the problem. For the studied $I_{eff}$ the variation of mass before and after the solving was of $0.4 \text{[cm}^2\text{]}$ in a surface of $0.08964 \text{[m}^2\text{]}$.

### 3.4 Fluid Flow modelization

#### 3.4.1 the $k$-$\epsilon$ and the $k$-$\omega$ solution

Comsol© operates for the magnetic physics with the A vector potential: in an axysimetrical study, I has only an azymuthal component, and so as consequence A. B has instead the $r$- and the $z$- component, so the Lorentz Force in differential form may be described as:

$$ F = J \times B = (J_\phi B_z) \hat{e}_r + (-J_\phi B_r) \hat{e}_z $$

and in a time domain analysis:

$$ F_r(t) = \frac{1}{2} \Re (I_\phi B_z) + \frac{1}{2} \Re (I_\phi B_z e^{j2\omega t}) $$

$$ F_z(t) = \frac{1}{2} \Re (I_\phi B_r) + \frac{1}{2} \Re (I_\phi B_r e^{j2\omega t}) $$

3.4. FLUID FLOW MODELIZATION

Comsol© provides internally the calculation of the Lorentz Force. Researches proved that the $2\omega t$ fluctuations of the (3.6) and (3.5) doesn’t affect significantly the velocity values \[12\]. The averaged volume force is applied to the domain, before and after the free surface shape meniscus modeling.

![Computed Lorentz Force](image)

Figure 3.10: Computed Lorentz Force with deformed and undeformed domain

At a certain point of the computational study, the value of Lorentz Force with the undeformed domain was wrongly calculated, because the mesh adopted for the magnetic and the fluid flow problem was the same and with boundary layers the prediction of the magnetic field was distorted. It’s important to separate mesh structures for each study, and keep in mind the purposes of the models. To lower also the errors, the interpolation function was chosen quadratic and not linear. The requested mesh for the fluid flow study was a breakthrough point. Since the the wall of the furnace are described with wall functions, the requested mesh may not be so complicated, even though it has not to be coarse and the boundary layers must cover little edge lengths of the wall. Looking
after the results, the structure of the boundary layer was important for two reasons, to give sufficient points for expressing the Lorentz Force on the domain, because the molecular friction happens really close to the surface and to permit the solver to compute the main vortices. From other CFD programs, like Fluent© or OpenFoam©, there’s the availability to check the boundary layers progress with giving a desiderable value of $y^+$. Comsol© doesn’t have this kind of procedure and makes the mesh automatically and with poor manual control: in the case different walls need different boundary layer densities, a manual set has to be made; for example, it decides the scale of the segment of each boundary layers along the wall and with a request of more refinement, these are compressed and stretched and especially they cover a minor region. In terms of quality, this means that boundary layers find a triangular element faster with an extremely fine mesh than with a normal one, and the gap between points suddenly increases, with a huge loss of quality. This study was first focused on creating a mesh for the SST $k$-$\omega$ that requires more attention on the different wall characteristics, so that $k$-$\epsilon$ and $k$-$\omega$ would have only benefit from it, because the resulted mesh was highly fine and with specific sets from the bottom and the lateral wall, but for the time requested by the solver in both time dependent and stationary study another way was chosen. In terms of definition, the more elements define the domain, the more the solution is accurate. For the second reason it is important also not to enlarge too much each element but to keep an edge length above $1/100$ of the fluid velocity, so for example 1 [m/s] corresponds to 1 [cm]. This is needed because when the gaps between points of the FEM simulation are too high and therefore too separated vortexes could disappear. The figures show solution for $k$-$\omega$ with a fine mesh and a coarse mesh:

![Figure 3.11: Turbulence description near the free surface for the coarse bounded and a fine mesh](image)

Another way to check the quality of the mesh, is to evaluate the distribution of the
turbulent viscosity on the element of the meshes\[1\] triangular mesh are often used in CFD to distribute points more fan than with a mapped mesh, and also to reduce the number of them in an area unit, without losing so much information. This works mostly for external flows, such as jets or even pipe flows, but with wall-bounded flows where the turbulence is given by a physic that operates inside the domain, nothing more than a mapped mesh was reliable to describe every parameter.

Figure 3.12: Eddy viscosity ratio for $k$-$\epsilon$ RANS solution with a bounded mesh

The use of bounded mesh dissolves the quantity of kinetic energy at the free surface shape, that results in a loss of consistent information. In the end, mapped mesh was the

\[1\] http://www.computationalfluiddynamics.com.au
structure that was chosen to figure out the problems. For solving the stationary RANS solution, a segregated solver with direct solvers has been automatically applied. The low kinematic viscosity $\mu$ of the fluid doesn’t permit to start computing directly: it’s unlikely that the low turbulent viscosity could be solved instantaneously. A parametric study was initialized with an high value of the viscosity, using a stationary solver to decrease the error: the same was applied to the next parametric step, turned to its maximum, and then recalculated for the lower kinematic viscosity. This manages to ensure convergence for the desired $\mu$. 
Figure 3.13: Eddy viscosity ratio for $k$-$\epsilon$ RANS solution with a full mapped mesh
3.4.2 the \( k-\epsilon \) solution

Working with \( k-\epsilon \) was the easiest way to find how vortexes are placed in the fluid flow domain: it is although not the best set that tries to predict the fluid flow. It is commonly used for the less effort one has to do with mesh structures, especially for a stationary study and for giving initialized conditions to other models like Low-Reynolds \( k-\epsilon \) or even the SST. Indeed, the more the mesh was fine, the less the solver process had to find solution for every kinematic viscosity parameter, but one must not forget that this also means higher time cost for a solution. The values that come from the \( k-\epsilon \) and the \( k-\omega \) are not necessary correct even if the solver had established convergence. Regarding to \( k-\epsilon \), it denotes problems on describing the dissipation rate when close to a wall: it’s a semi-empirical formula (Verseeg, Malalasekera et al. 2005) and the closure coefficients are balanced to match results for decay of isotropic turbulence but for the MHD application they seem not.

Experiments conducted at ETP [12] show high dissipations that happen near the wall and cause high mixing in the zone. Since the presence of the wall functions, these are ignored and flow development is uncertain. For solving heat exchanges between the fluid and the wall, \( k-\epsilon \) seems not the most suitable formula. The graphics on (3.14) and (3.15) interests the stationary solution: these represent the behavior of the \( k-\epsilon \) with the deformed mesh and with the rectangular domain; the comparison was interesting to evaluate how the deformation could affect, or not, comprehensively. Comparing the velocities obtained, there aren’t reasonable changes, and \( k-\epsilon \) seems reliable on working with deformed mesh. Checking the distribution of the kinetic energy, show that the value of kinetic energy computed in the middle of the bottom vortex for a deformed domain is lower than the one obtained in the rectangular one. There’s anyway a high value of it on the top corner of the fluid domain, where no vortexes are created, and one has to bear in mind that the \( k-\epsilon \) predicts the kinetic energy to maximize at the center of the vortexes [8].

After that it’s possible to compare also the figure (3.13) with the figure (3.18) to see how the value of the turbulent viscosity ratio changes its displacement in the presence of the curved surface. In this case, an inconsistent value of the turbulent viscosity ratio appears at the free surface shape: in both cases, the distribution appear a bit debatable. The turbulent viscosity is directly related to the kinetic energy and the reciprocal of the dissipation rate but the latter has a value only on the fluid corner, due to errors at the connection point between the free surface shape and the wall. For the rest of the meniscus, this does not change and \( k \) suffers of the slip condition, and so as consequence \( \mu_t \). Axial velocities show how much this variation compromises velocities, where indeed the bottom vortex has also a modification after the meniscus shape application, but for the top side this means a complete different concept.
Figure 3.14: $k$-$\epsilon$ Vortices with velocity profile the deformed mesh
Figure 3.15: $k$-$\epsilon$ Vortexes with velocity profile without the deformed mesh
3.4. FLUID FLOW MODELIZATION

Figure 3.16: $k$-$\epsilon$ turbulent kinetic energy without the deformed mesh
Figure 3.17: $k$-$\epsilon$ turbulent kinetic energy with the deformed mesh
Figure 3.18: Eddy viscosity ratio without the deformed mesh
Figure 3.19: Comparison of axial velocities for the $k$-$\epsilon$ solution at different radius
3.4.3 the \( k-\omega \) solution

\( k-\omega \) allows for a more accurate near wall treatment with an automatic switch from a wall function to a low-Reynolds number formulation based on grid spacing, and this helps to compute more correctly the turbulent viscosity on the fluid domain: However this refinement on computation needs a good mesh structure.

In this case, an unique kind of structure for \( k-\omega \) and \( k-\epsilon \) permits to solve both models without loosing integrity on walls, but that’s a case [1]: with lower kinematic viscosities, for the \( \omega \) transport equation has higher request than the \( \epsilon \) one, but that’s something it can be conjured with Comsol\textsuperscript{©} whether with several trials.

Solver procedure and check of the results are equal to the \( k-\epsilon \) problem; in this case, there are no visible changes in vortexes and velocity value distribution

\( k-\omega \) appear more accurate and the accuracy at the free surface shape is even acceptable, and the boundary slip condition does not involve too much errors because the kinetic energy is better predicted.

It’s evaluable, comparing with the \( k-\epsilon \) procedure, that the deformed mesh has not a big influence for the vortex it has to form on the upper part, even this deformation is not so preponderant. The distortion of turbulent kinetic energy on the corner energy appears also in the \( k-\omega \) model, but here one can acknowledge more than in the \( k-\epsilon \) that is an error of the computation instead of a real value. The change of distribution is not so huge and remains similar to the case of a rectangular domain, something that didn’t happen on the \( k-\epsilon \) case.
Figure 3.20: $k$-$\omega$ Vortices with velocity profile without the deformed mesh
Figure 3.21: $k$-$\omega$ Vortices with velocity profile with the deformed mesh
Figure 3.22: $k$-$\omega$ turbulent kinetic energy without the deformed mesh
Figure 3.23: $k$-$\omega$ turbulent kinetic energy with the deformed mesh
Figure 3.24: $k$-$\omega$ eddy viscosity ratio without the deformed mesh
Figure 3.25: $k$-$\omega$ eddy viscosity ratio with the deformed mesh
3.4. FLUID FLOW MODELIZATION

![Diagram of velocity profile at 0 cm and 2.5 cm depths, showing deformed and undeformed mesh compared to the depth of the furnace in millimeters.](image)
Figure 3.26: Comparison of axial velocities for the $k$-$\omega$ solution at different radius
3.4.4 The Low-Re $k-\epsilon$ solution

Mesh requests for this method are not higher as one supposes for a Low-Re model: it’s true that solves the fluid flow for the entire region, but there’s not a limit operator that sets the value of $\epsilon$ as for a SST model for $\omega$.

The mesh adopted shall continue to be finer as possible, but with necessary wall refinement for the buffer layer and viscous sublayer.

Anyway, stationary solvers presented difficulty in getting convergence, so the time dependent solver was applied and the time studied was of $500$ [s], in order to reach a stationary solution alike.

There was no need to initialize values of velocity that was more needed in case of higher Lorentz Forces. In comparison with the $k-\epsilon$ solution, the behavior near to the free surface shape is completely different and shows how this method is not affected by the the deformed mesh in correspondence to a slip boundary condition. Like the $k-\omega$ model, there’s a very low modification on velocity profile between the two proceedings, even if the shape of velocity line graphic is different: velocity profile closer to the wall shows that Low Reynolds $k-\epsilon$ results smoother than the $k-\omega$ which presents a swelling over the bottom vortex velocity line.
Figure 3.27: Low Reynolds $k$-$\epsilon$ Vortexes with velocity profile without the deformed mesh
Figure 3.28: Low Reynolds $k$-$\epsilon$ Vortices with velocity profile with the deformed mesh
Figure 3.29: The kinetic Low Reynolds $k$-$\epsilon$ energy without the deformed mesh
Figure 3.30: The kinetic Low Reynolds $k$-$\epsilon$ energy with the deformed mesh.
Figure 3.31: The Low Reynolds $k$-$\epsilon$ eddy viscosity ratio without the deformed mesh
Figure 3.32: The Low Reynolds $k$-$\epsilon$ eddy viscosity ratio with the deformed mesh
CHAPTER 3. NUMERICAL SIMULATION

velocity profile at 0 cm

velocity profile at 2.5 cm
Figure 3.33: Comparison of axial velocities for the Low Reynolds $k$-$\epsilon$ solution at different radius
4. Measurements

The Ultrasound Doppler velocimetry is a measure instrument that manages to measures velocity profile in real time in various liquids, obtaining informations from particles moving in the fluid hit by the ultrasonic field and reflecting echoes. Its use is going to increase due to his reliability on several kind of fluids, such as water, mud, fluid with particles like dust or bubbles, opaque liquids. On the metallurgical applications, his applications are relevant but more studies must be done, comparing for example the velocity profile with other resources like the power spectrum and flow rate with more transducers. The measurement were held using the pulsed UD V that has the capability to define spatial informations related to the velocity values in real time. The main advantages are the fact that it is a non invasive instrument, and can be put outside the fluid flow since the beam cross any wall that enclose the fluid flow.

4.1 The pulsed Ultrasound Doppler Velocimetry

The principle of the pUDV is based on the Doppler Effect, that is the shift of an acoustic or electromagnetic wave whenever the source or the receiver are moving between each other. From the side of the receiver, the frequency is given by:

\[ f_R = \left( \frac{c - v_R}{c - v_S} \right) f_C \]  \hspace{1cm} (4.1)

with \( f_C \) the emitted frequency, \( v_S \) the velocity of the emitter, \( v_R \) of the receiver, \( c \) the speed of the wave propagation. For a moment let’s say that is considered negative the velocity \( v \) when the receiver is going toward the emitter, so that it could be written the Doppler shift frequency:

\[ f_D = \left( \frac{c - v_R}{c - v_S} - 1 \right) f_C \]  \hspace{1cm} (4.2)

The pUDV has emitter and receiver fixed in space \((v_S = -v_R = v)\), and studies moving elements which are hit by the ultrasonic wave. The reflection is the concept of how it works: the emitter sends an impulse that the element, when moving, reflects changing the direction of the wave and the returned one is catch by the receiver that believes this new
wave as something generated from a source from a distance equal to the total distance traveled by the wave. This proceeding gives the opportunity to evaluate both velocity and distance informations. It’s possible to imagine emitter, receiver and reflecting element moving apart on a same line with identical velocities, so the receiver reads the shift:

\[ f_D = -\left(\frac{2v}{c + v}\right) f_c \]  \hspace{1cm} (4.3)

and in the case \( c >> v \):

\[ v = -\frac{f_D c}{2f_C} \]  \hspace{1cm} (4.4)

that is the equation reflector velocity. Our reality considers particles that don’t all go along the direction of the ultrasonic beam but their movement form an angle \( \theta_1 \) in correspondence of the ultrasonic beam propagation, the equation (4.4) has to be modified:

\[ v = -\frac{f_D c}{2f_C \cos(\theta_1)} \]  \hspace{1cm} (4.5)

The sign of \( f_D \) is not important as the Doppler shift detector is sensitive only to the magnitude of it. In the case that emitter and receiver are the same component (the transducer) and the moving element moves on a medium with the velocity \( v \) and the direction defined by \( \theta_1 \), the latter perceives a frequency of the wave given by:

\[ f_t = f_E - \frac{f_C \cos(\theta_1)}{c} \]  \hspace{1cm} (4.6)

with \( f_E \) the emitter/transducer frequency. In contrast to the continuous UDV, pUDV doesn’t send unique continuous waves but ultrasonic wave packages, periodically, and the receiver acquires the echoes coming from the particles that cross the path of the ultrasonic beam. These echoes are accepted by the transducer for a short period of time, in order to associate the emission of the bursts to the proper acquisition, and this is possible using an internal operator-adjustable delay and related to the fixed pulse repetition frequency \( P_{rf} \). The receiver acquires the incoming echoes from all the particles at the same time and measure the shift of positions of the scatterers. The measurement system compute the phase shift between the pulses received to add information such as the distance of the particles \( P \). The number of wavelengths contained in the two way path in between the transducer and the target is \( 2P/\lambda \). Because for a wavelength there’s an angular excursion of \( 2\pi \) radians, the total excursion \( \phi \) will be \( 4\pi P/\lambda \). During the motion of the element, both \( P \) and \( \phi \) are simultaneously changing. What is important of \( \phi \) is that its change in time corresponds to a frequency, and more precisely the doppler angular frequency \( \omega_D \):

\[ \omega_D = 2\pi f_D = \frac{d\phi}{dt} = \frac{4\pi}{\lambda} \frac{dP}{dt} = \frac{4\pi v}{\lambda} \]  \hspace{1cm} (4.7)
4.1. THE PULSED ULTRASOUND DOPPLER VELOCIMETRY

As a consequence the Doppler frequency is as follows:

\[ f_D = \frac{2\nu}{\lambda} = \frac{2\nu f_C}{c} \]  

(4.8)

From (4.8) is it possible to find the distance for the target:

\[ P = \frac{cT_D}{2} \]  

(4.9)

with the previous considerations (4.5) the velocity of the target can be measured by variation in depth of it in two different emission bursts, defined by \( T_{Prf} \):

\[ (P_2 - P_1) = \Delta \cos (\theta) = \nu T_{Prf} \cos (\theta) = \frac{c}{2} (T_2 - T_1) \]  

(4.10)

![Figure 4.1: Description of the acquisition (19)](image)

Because the term \( T_2 - T_1 \) is always really short (around [\( \mu s \)]), it is convenient to replace it with the measurement of the phase shift of the two distinct echoes:

\[ \delta = \omega t = 2\pi f_C (T_2 - T_1) \]  

(4.11)

and therefore:

\[ v = \frac{c(T_2 - T_1)}{2T_{Prf} \cos (\theta)} \]  

(4.12)

It results similar to (4.4) but has different meaning: velocities are obtained from shifts between position whereas in (4.4) they’re derived finding the Doppler shift frequency in the received signal. To say it more properly, the distinction consists on the evaluation of the shift in position of the scatterers and and not the shift in emitted frequency, what the whole system analyzes. The Doppler effect participates in one of the affections of the results: the transmitted wave could appear compressed or stretched. Another problem is that the target move itself from the transducer and the received signals undergo a progressive time shift with respect to the time of the transmission. The transducer, with its piezoelectric crystal, operates either as an emitter and a receiver: the electrical
circuit is shared for the two functions, and so the problem of alignment is avoided. It is acoustically and electrically matched to the input impedance and that permits to have bursts that conform the needed electrical excitation. The piezoelectric is excited by short high-voltage unipolar circuits, and vibrates in its resonance thickness expander mode. The echoes are converted even in the crystal, and sent to the reception amplifier that belongs to the Signal Bloc. The amplification is made within two processes:

- an overall gain that is independent from the quality and kind of signal
- a time dependent gain that check the signal according to the depth (related to the Time Gain Control) and return it higher because deeper waves are eased off in their propagation.

The TGC is important in relation with the frequency of the emission chosen for the measurements: when using high frequencies, its slope has to be steeper. Increasing the logarithmic amplifier gain in synchrony with the arrival time of echoes provides a simple means of compression by approximately compensating for attenuation [15].

4.2 The Ultrasonic Field

The transducer provides by Signal-Processing© has different diameters and maximal frequency response: it’s not only a matter of size, whenever little pipe flows must be studied or other kind of applications: the ultrasonic burst packet emitted by the transducer is related to the two cited dimensions. Therefore the size and shape of this sample volume determines the flow meter sensitivity and accuracy. What the Signal Block consequently does with the velocity profile are weighted average of the ultrasonic intensity and the flow field velocity over the sample volume. The focused beam shape changes in its propagation and the distance from the transducer indicates how the geometry of the sample volume varies. Using Huygen’s principle, looking at the ultrasonic single-element transducer as a combination of several adjacent point sources, each generating a spherical wave, the shape of the ultrasonic field is illustrated in the figure:

![Figure 4.2: The Ultrasonic Field](image)

It is provided an equation [11] for acoustic fields created by piston-like emitter (equal...
4.2. **THE ULTRASONIC FIELD**

as a normal sound house station). The intensity of the acoustic field is as follows:

\[
\frac{I_z}{I_0} = \sin \left[ \frac{\pi}{\lambda} \left( \sqrt{a^2 - x^2} - x \right) \right]
\]  
(4.13)

with \(a\) the radius at a determined height of the axis \(x\) and \(\lambda\) the wavelength. The propagation has primary a near field zone, the acoustic field is nearly cylindrical, with a diameter slightly less than the diameter of the emitter. In this region the intensity of the acoustic waves oscillates and the echo amplitude goes through a series of maxima and minima. For liquids like water, these oscillations do not interfere because they’re much smaller than the dimension of the measured volumes, but for opaque liquid, it is common to start measuring after 5-6 cm. The length of the far field is given by:

\[
L_{NF} = \frac{4d^2 - \lambda^2}{4\lambda}
\]  
(4.14)

where \(d\) is the diameter of the transducer. After this region, the intensity of the acoustic waves along the axis decreases as the inverse of the square of the distance from the transducer (according to the inverse-square law for point sources). Small oscillations usually appear in the radial direction, normal to the axis of propagation, like in the figure.

![Figure 4.3: Oscillation related to the Field Regions](image)

Most of the acoustic energy is contained in a cone. The relationship between the acoustic field intensity and the angle of the transducer axis depends on:

\[
D_r (\gamma) = \frac{2J_1 (ka \sin (\gamma))}{ka \sin (\gamma)}
\]  
(4.15)

that is the directivity function, with \(J_1\) the first order Bessel function of the first grade, and \(k = 2\pi/\lambda\) the wave number. The frequency that the operator can choose has a strong relation with the maximum velocity that the transducer can read by its crystal:

\[
v_{max} = \frac{c}{4f_C \cos (\theta_1) T_{prf}}
\]  
(4.16)
In the case the measured velocity is higher than this maximum, the aliasing phenomenon appear: this situation come from the Nyquist Theorem and the consequence is that the Doppler frequencies above the half of the sampling frequency $1/T_{prf}$ are are folded back in the low frequency region; on the other hand a higher frequency gives a better axial resolution but also causes higher attenuation of the ultrasonic waves.

Consecutively received signals are shifted in time compared with the proceeding and the precedent sample volume as a result of motion of the scatterers. This shift involves a progressive change in the phase relationship between the ultrasound signal packet and the master oscillator, which is exactly what the pulsed ultrasound doppler velocimetry detects. Whenever it’s chosen to increase the gap time, the time delay of the the received signals also increases between two scatters, hence the distance between the transducer and the scattering particle arises. Two received signals are compared in order to analyze this situation. Pulses are emitted after a delay of $T_{prf}$ seconds. In the case echoes present the same frequency, this means no movement took place; if not, time delay or corresponding phase shift is acquired between consecutive emissions. A fact that may compromise the results is that the received signal is not only translated in time from pulse to pulse, but does also change shape due to the construction of the signal from the various responses from the scatterers that belong to the sample volume with different velocities. It’s possible to say [15] that constructive and destructive interference happen. Scatterers move at different velocities and their relative position changes over time, modifying the interference between them. Combined with the Doppler effect, this artifacts can disturb the complete solution.

4.3 Comparison of measurements and computed velocities

The experiments at ETP were taken twice, using a filling level of 536 mm and acquiring values of the velocity in seven sample from the center to the lateral wall, each placed
4.3. COMPARISON OF MEASUREMENTS AND COMPUTED VELOCITIES

at a distance of 2.5 cm from the latter. This was decided because the probe must be positioned vertically and parallel to the axes of the furnace cylinder, so that angle $\theta_1 = 0$ and, due to the cumbersome mechanical holder and the problem of incurring wave diffractions from the wall, the maximum radius belonging to a sample was of 15 cm. Previous experiments, matched with other measurements technique like the potential probe, show wrong acquisitions in a range of 10 cm from the probe. This couldn’t help to find how much an undeformed mesh can affect the results of velocity in measurements, whereas the control of the turbulent kinetic energy is something that regards particularly the potential probe [16]. Acquisition and comparison were only focused on the lower vortex, with a height of 376 cm from the bottom. The graphics presented link the measurements with the computation of the various model with deformed and undeformed mesh; below it is shown the $k-\epsilon$ comparison.
4.3. COMPARISON OF MEASUREMENTS AND COMPUTED VELOCITIES

**Velocity Profile at 7.5 cm**

- Measurements data
- Deformed mesh
- Undeformed mesh

**Velocity Profile at 10 cm**

- Measurements data
- Deformed mesh
- Undeformed mesh
The matching is acceptable until the velocity field belongs to the free shear flow: indeed, after a radius of 10 [cm], representation of velocities are not similar any more, with differences both in peak value and velocity distribution. Examining the deformed mesh and the undeformed mesh solution, big differences are yielded in velocity shape: the height of 376 [cm] corresponds to height where problems at the top vortex appear. The rectangular undeformed mesh seems more accurate in peak mean values and velocity profile, as it could be seen from the attacks of the profiles. The next case concerns the $k$-$\omega$ study.
4.3. COMPARISON OF MEASUREMENTS AND COMPUTED VELOCITIES

![Velocity profile at 5 cm](image1)

![Velocity profile at 7.5 cm](image2)
In the free shear flow $K-\omega$ shows good correspondence at the attacks of the velocity profile, and that means a good correspondence in the length scale of the vortices, but it has the worst behavior in computing velocity: the velocity maximums are much higher than the one computed in the $k-\epsilon$ solution, and even the velocity profile results not accurate for the free shear flow. The swelling distorts the velocity graph comprehensively.
CHAPTER 4. MEASUREMENTS

velocity profile at 0 cm

velocity profile at 2.5 cm
4.3. COMPARISON OF MEASUREMENTS AND COMPUTED VELOCITIES

**Velocity Profile at 5 cm**

- Measurements data
- Deformed mesh
- Undeformed mesh

**Velocity Profile at 7.5 cm**

- Measurements data
- Deformed mesh
- Undeformed mesh
4.3. COMPARISON OF MEASUREMENTS AND COMPUTED VELOCITIES

The Low-Reynolds $k$-$\epsilon$ model seems to describe more accurately the fluid flow evolution, for the free shear flow. It shows good comparison between the peak mean velocities also for the value placed at 7.5 [cm] from the center, even if the velocity curves are a bit displaced between each other. An acceptable description of velocity appears also at the distance 15 [cm], where the initial evolution of the curve seems similar.

Figure 4.6: Comparison of axial velocities for the Low Reynolds $k$-$\epsilon$ solution at different radius
5. Conclusions

The aim of this work was to develop the most correct modelization of the fluid flow. Preliminary models combining deformation of the free surface and fluid flow calculations have been tested only with RANS approximation. In the case of $k$-$\epsilon$, the proximity of a curved surface enhances the poor definition of kinetic energy and eddy viscosity, already precluded for the bad approximation on the slip boundary condition. In terms of velocities, this means a huge change of scale of vortexes and velocity representation. $k$-$\omega$ doesn’t suffer of this problem, and gives a bit more precise computation of the kinetic energy, but the distribution of velocity appear much higher than the velocities acquired from the measurements. All these models present bad quality representation of what happens near to the wall, due to the absence of mixing viscous region near the wall. Low Reynolds $k$-$\epsilon$, even if it can’t give a precise description of mixing viscous region in the nearby of the walls, thanks to the damping functions permits to describe more accurately the fluid flow evolution, also when reaching the wall. As a conclusion of this study, it seems the most suitable model for calculating velocities on an induction furnace. SST requires more attention than the other models in particular it requires a fine mesh to achieve satisfactory result. After many attempts in order to achieve the SST solution without lack of definition and with a mesh structure that was not computationally expensive, I had to opt for a full mapped mesh, more accurate but time consuming. The stationary solver, like for the Low Reynolds $k$-$\epsilon$ model, did not reach convergence in a reasonable time. Several trials with iterative domain decomposition solvers or multigrid instance, were carried out obtaining always a divergence of the solution. A bad definition changes all the behavior of the fluid flow. Due to time limits, the trials with SST approach were stopped. The comparison of the fluid flow with the UDV measurements shows that the Low Reynolds $k$-$\epsilon$ gives better results among the other studies. The Low-Reynolds $k$-$\epsilon$ model seems to describe more accurately the fluid flow evolution, without lack of definition when reaching the lateral wall. In any graph the peak mean value is higher, but this may result in the approximation applied for the computational study.
A. Integrative Formulas

A.1 Integration function

Considering the figure (3.4), an easy way to define $f(x)$ was to adopt the Langrangian polynomial interpolation:

$$l_k(x) = \prod_{i=0,i\neq k}^{n} \frac{x-x_i}{x_k-x_i} \quad (A.1)$$

with $k = 0, ..., n$. With $n$-nodes, the obtained polynomial will be $n-1$:

$$P(x) = y_0l_0 + y_1l_1 + y_2l_2 + \ldots + y_nl_n \quad (A.2)$$

The set of figure (3.4) has three points, and that means a computation of a second order equation:

$$y_0 = z_0 \quad y_1 = z_1 \quad y_2 = z_2$$

$$l_0 = \frac{x-x_1}{x_0-x_1} \cdot \frac{x-x_2}{x_0-x_2} \quad l_1 = \frac{x-x_0}{x_1-x_0} \cdot \frac{x-x_2}{x_1-x_2} \quad l_2 = \frac{x-x_0}{x_2-x_0} \cdot \frac{x-x_1}{x_2-x_1}$$

When $x_0 = r_0 = 0$, the resolution of the equation (3.3) leads to the solution:

$$\int P(x)dx = \frac{z_0}{r_1r_2} \cdot \left(-\frac{1}{6}r_2^2 - \frac{3}{2}r_1r_2\right) + \frac{z_1}{r_1^2 - r_1r_2} \left(-\frac{1}{6}r_2^3\right) + \frac{z_2}{r_2^2 - r_2r_1} \cdot \left(\frac{1}{3}r_2^2 - \frac{1}{2}r_2^2r_1\right)$$

A.2 Curvilinear description of the Lorentz Force

As mentioned in (), the first term on the right side of the equation could be interpreted as a surface stress and integrated for evaluating the effect of a distributed body force. Another way is to represent $(\mathbf{B} \cdot \nabla)\mathbf{B}$ in terms of curvilinear coordinates attached to a streamline:

$$(\mathbf{B} \cdot \nabla)\mathbf{B} = B\frac{\partial B}{\partial s}\hat{e}_t - \frac{B^2}{R}\hat{e}_t \quad (A.3)$$
with $B$ as the norm of $\mathbf{B}$ and $R$ is the local radius of curvature of the field line. As a consequence:

$$\mathbf{J} \times \mathbf{B} = \frac{\partial}{\partial s} \left[ \frac{B^2}{2\mu} \right] \hat{e}_t - \frac{B^2}{R\mu} \hat{e}_n - \nabla \left( \frac{B^2}{2\mu} \right)$$  \hspace{1cm} (A.4)
List of Figures
Bibliography


