CFD MODELLING OF HEAT TRANSFER IN BLAST FURNACE

A thesis submitted in partial fulfilment of the

requirement for the degree of

Bachelor of Technology

In

Metallurgical and Materials Engineering

By

BISWAJIT DALAI (111MM0268)

DEPARTMENT OF METALLURGICAL & MATERIALS ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA

2015

CFD MODELLING OF HEAT TRANSFER IN BLAST FURNACE

A thesis submitted in partial fulfilment of the

requirement for the degree of

Bachelor of Technology

In

Metallurgical and Materials Engineering

By

BISWAJIT DALAI (111MM0268)

DEPARTMENT OF METALLURGICAL & MATERIALS ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA

2015

NATIONAL INSTITUTE OF TECHNOLOGY

ROURKELA

<u>CERTIFICATE</u>

This is to certify that the work in this project report entitled "**CFD Modelling of Heat Transfer in Blast Furnace**" by **BISWAJIT DALAI (111MM0268)** has been carried out under my supervision in partial fulfilment of the requirements for the degree of **Bachelor of Technology** in **Metallurgical and Materials engineering, National Institute of Technology, Rourkela** and is an authentic work carried out by them under my guidance.

To the best of my knowledge, this work has not been submitted to any other University/Institute for the award of any degree or diploma.

Place:
Date:

Dr. Natraj Yedla
Department of Metallurgical and Materials engineering
National Institute of Technology Rourkela, Orissa
769008

ACKNOWLEDGEMENTS

I owe great many thanks to a lot of people who helped me and supported me during my undergraduate project work for the partial fulfilment of Bachelor in Technology degree.

I would like to convey my sincere gratitude to Asst. Prof. Dr. Natraj Yedla, my guide and supervisor, for guiding me all through the course of time with attention and care. He has always come forward to offer his help in the time of need. Without his constant motivation, it would have been very difficult for me to complete the project work quite successfully.

My deepest thanks to Associate Prof. Dr. Smarajit Sarkar and Associate Prof. Dr. Anindya Basu, the course co-ordinators, for their help and advice during the course of my project work.

I am highly indebted to Asst. Prof. Dr. Snehanshu Pal. Thanks to his proper guidance and invaluable advice; I was able to overcome many problems during the period of work.

I would also like to thank Mr Rahul Kumar, M. Tech Scholar of our Department, for sharing his knowledge in Computational Fluid Dynamics with utter patience.

My sincere thanks to Mr Yugesh Patnaik and Mr.Prabir Kumar Patra, my fellow batch-mates from Mechanical Department, who were always there to help me out with the problems I faced in using ANSYS 15.0, even in the weirdest hours.

Last but not the least; I would like to extend my regards to all my fellow classmates for their help and support.

Place:

Date:

Biswajit Dalai (111MM0268)

TABLE OF CONTENTS

LIST OF FIGURES

ABSTRACT

Iron blast furnace is used in the metallurgical field to extract molten pig iron from its ore through a reduction mechanism. The furnace is a vertical shaft with circular cross section. It has five main parts: stack, belly, bosh, tuyeres and hearth. Amongst these regions, hearth is the most important one for the asset life of a furnace. Erosion of refractory lining of the hearth reduces the furnace's campaign life. So it is necessary to understand the interactions occurring between the slag, molten metal and the refractories. But the severe operating conditions and very high temperature inside the hearth make it impossible to practically observe the processes taking place within it. In order to overcome this problem, the hearth is modelled by using various Computational Fluid Dynamics (CFD) soft-wares such as ANSYS Fluent, ANSYS-CFX, FLUENT for CATIA V5, ANSYS CFD-Flo etc. The numerical model is then supplied with data which are already known from practical situations as boundary conditions. Proper physical properties of the materials are also used as input. The software runs several simulations and provides us with the result that can validate the experimental observations up to the most accurate level. In this study, temperature distribution profile inside a blast furnace hearth has been shown by modelling a simple hearth with the help of ANSYS 15.0 Workbench. The model is simulated by changing some parameters and making several assumptions. The discrepancy in the calculated and the observed temperature opens up new scope for further improvement.

Keywords: Blast furnace hearth, campaign life, Computational Fluid Dynamics (CFD), ANSYS, simulations.

CHAPTER 1

INTRODUCTION

1.1. Blast Furnace:

Blast furnace is a large furnace which is used in metallurgical field to produce metals, especially pig iron. Lead or copper can also be produced in the blast furnace. But our study is confined to the extraction of iron from its ore.

1.1.1. Various regions within an iron blast furnace:

Blast furnace is a vertical shaft with circular cross section. It is composed of five main parts [1]. They are:

i)   Hearth- a crucible-shaped portion which is present at the bottom of the furnace;

ii)   Stack- a vertical shaft that extends from stock line to mantle level;

iii) Belly- the cylindrical portion below the stack;

iv)   Bosh- an intermediate zone between the hearth and the belly;

v)   Tuyeres- series of pipes located around the circumference through which hot air blast is blown into the furnace.

The schematic diagram of an iron blast furnace is shown in the figure below [1].
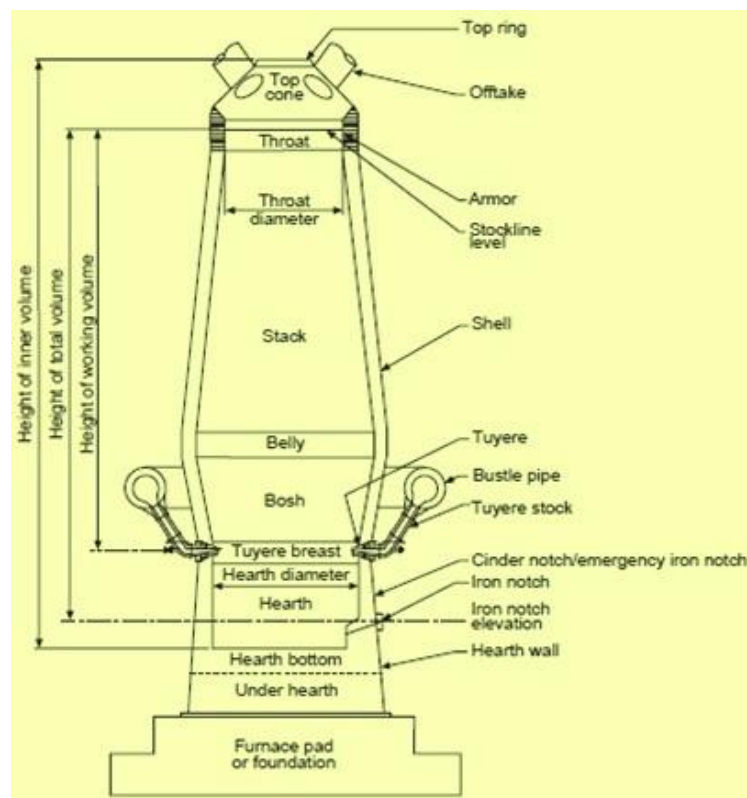
FIG 1: *Schematic diagram of an iron blast furnace*

1.1.2. Functions inside an iron blast furnace:

Blast furnace is a counter current heat exchanger. The charge is supplied from the top of the blast furnace at room temperature. It consists of fuel (coke), iron ore (haematite) pellets or sinter and flux (limestone). The interior is filled with alternating layers of coke and ore [2]. The fed coke moves down to the hearth of the blast furnace and constructs a porous bed of un-burnt coke called dead-man [3]. Meanwhile, a hot blast of air which has been preheated to 900-1250° C is introduced into the lower section through the tuyeres. As a result, a cavity known as raceway is formed around the exit of a tuyere. In the raceway, the preheated air reacts with the dead-man coke to generate heat and reducing gases. These gases and heat are used to produce hot metal and molten slag. Due to its greater density, molten iron sits at the bottom of the hearth. Slag floats over the hot metal since it has comparatively lower density. The hot metal and the molten slag are tapped through the tap hole and the slag notch from time to time [1]. The top gas which is also known as blast furnace gas exits from the top of the furnace.

1.1.3. Reactions inside the blast furnace:

Blast furnace reactions take place in different regions [1]. They are:
i)   The upper region, i.e. Stack or Shaft
ii) The middle region, i.e. Belly and Bosh
iii) The lower region, i.e. Hearth

The major reactions inside a blast furnace can be classified into the following categories [1]:
i)   Removal of moisture from the raw materials
$$H_2O + C = H_2 + CO$$
ii) Reduction of iron oxides by CO
$$3Fe_2O_3 + CO = 2Fe_3O_4 + CO_2$$
$$Fe_3O_4 + CO = 3Feo + CO_2$$
iii) Gasification of carbon by CO_2
$$C + CO_2 = 2CO$$
iv) Dissociation of $CaCO_3$
$$CaCO_3 = CaO + CO_2$$
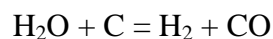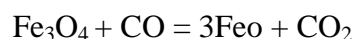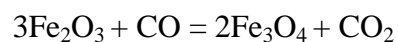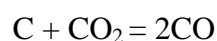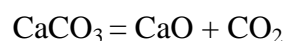
v) Reduction of FeO by carbon:

$$FeO + C = Fe + CO$$
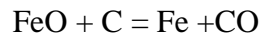
vi) Reduction of some other oxides of ore by carbon

$$SiO_2 + 2C = Si + 2CO$$

$$MnO + C = Mn + CO$$

$$P_2O_5 + 5C = 2P + 5CO$$

$$S + (O) + C = (S) + CO$$
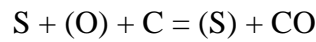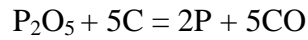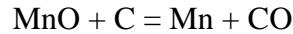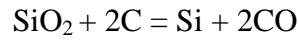
vii) Combustion of coke and coal in front of tuyeres

$$2C + O_2 = 2CO$$
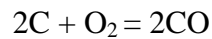
1.1.4. Temperature distribution inside a furnace:

The charge is supplied from the top of the blast furnace at room temperature [1]. The temperature in the stack region increases from 200° C (500 K) at the stock line level to 1100-1200° C (1400-1500 K) at the mantle level. The range of temperature in the belly region is between 1400-1600° C (1700-1900 K). The highest temperature zone of the blast furnace is at the level of tuyeres where temperature is around 1900-2000° C (2200-2300 K). Below the tuyeres level, the temperature decreases gradually from 1600° C (1900 K) to 1450° C (1700 K) down the hearth region.

The following figure shows the temperature distribution inside an iron blast furnace.
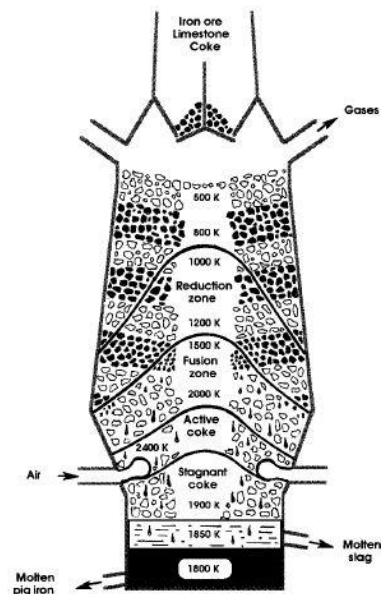
FIG 2: *Temperature distribution within an iron blast furnace*

1.2. Modelling and simulation:

Modelling and simulation is getting information about how something will behave without actually testing it in real life. Representing the real systems either via physical reproductions at smaller scale, or via mathematical models that allow representing the dynamics of the system via simulation, allows exploring system behaviour in an articulated way which is often either not possible or too risky in the real world.

1.2.1. Model:

A model is a simplified representation of a system at some particular point in time or space intended to promote understanding of the real system.

1.2.2. Simulation:

A simulation is the manipulation of a model in such a way that it operates on time or space to compress it. Thus one can perceive the interactions that would not otherwise be apparent because of their separation in time or space.

1.3. Computational Fluid Dynamics:

Computational Fluid Dynamics is usually abbreviated as CFD. Fluid (gas and liquid) flows are governed by Partial Differential Equations (PDE) which represent conservation of the mass, momentum and energy. CFD is a branch of fluid mechanics which functions by replacing such PDE systems by a set of algebraic equations that can be solved using digital computers. CFD provides a qualitative (sometimes quantitative) prediction of fluid flows by means of a) Mathematical modelling, b) Numerical methods and c) Software tools.

1.3.1. Functioning of CFD modelling:

There are five main steps of a CFD design cycle:

i) Analyst states the problem statement that is to be solved by using CFD.
ii) A mathematical model is prepared and boundary conditions are given.
iii) The computer code (software) embodies these conditions and instructions (algorithms) in details.
iv) Computer hard-wares perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions.
v) The user observes the animations and interprets the simulation results.

1.3.2. Advantages of CFD modelling:

Some of the advantages of CFD modelling are listed below:

i) CFD gives an insight into flow patterns that are difficult, expensive or impossible to study using traditional experimental techniques.

ii) CFD software is portable, easy to use and modify.

iii) It does not replace the measurements completely but the amount of experimentation and the overall cost can be significantly reduced.

iv) CFD simulations are cheaper, faster and can be used for multiple purposes.

1.3.3. Limitations of CFD modelling:

The results of a CFD simulation are never 100% reliable because:

i) The input data may involve too much guess work or imprecision.

ii) The mathematical model of the problem available at hand may not be adequate.

iii) The accuracy of the results is limited by the available computing power.

1.3.4. Applications of CFD modelling:

CFD is used in various fields such as:

i) Architects design safe and comfortable living environments.

ii) Aerodynamics characteristics of vehicles are improved by the designers.

iii) Surgeons use computational hemodynamic to cure arterial diseases.

iv) Meteorologists are able to forecast the weather condition and warn the people about natural disasters.

v) Risks arising from radiation and other hazards can be minimized by safety experts.

vi) Military forces develop safety weapons and estimate the amount of damage they can control.

vii) People with artistic taste can profit by selling colourful pictures.

1.3.5. CFD Soft-wares:

Some of the CFD soft-wares used for modelling and simulation are a) ANSYS Fluent, b) ANSYS-CFX, c) FLUENT for CATIA V5, d) ANSYS CFD-Flo, e) STAR-CD, f) FEMLAB, g) FEATFLOW etc.

1.3.5.1. ANSYS CFD Software:

ANSYS CFD allows us to predict the impact of fluid flows on our test systems by simulating in a virtual environment. It considers design and manufacturing and the end use as well. The software's unparalleled fluid flow analysis capabilities can be used to design and optimize new equipment and to troubleshoot already existing problems.

ANSYS CFD solutions are fully integrated into the ANSYS Workbench platform. This environment delivers high productivity and easy-to-use workflows. Workbench integrates all our workflow needs as well as multi-physics functionality. It has broadly physical modelling capabilities such as to model: a) Flow behaviour, b) Turbulence, c) Heat transfer etc.

Industrial applications of this software include: a) Combustion in furnace, b) In-cylinder combustion, c) Waste water treatment plants etc.

1.4. Modelling of blast furnace hearth:

1.4.1. Geometry of hearth:

The following figure shows the simple diagram of a blast furnace hearth made of different materials.
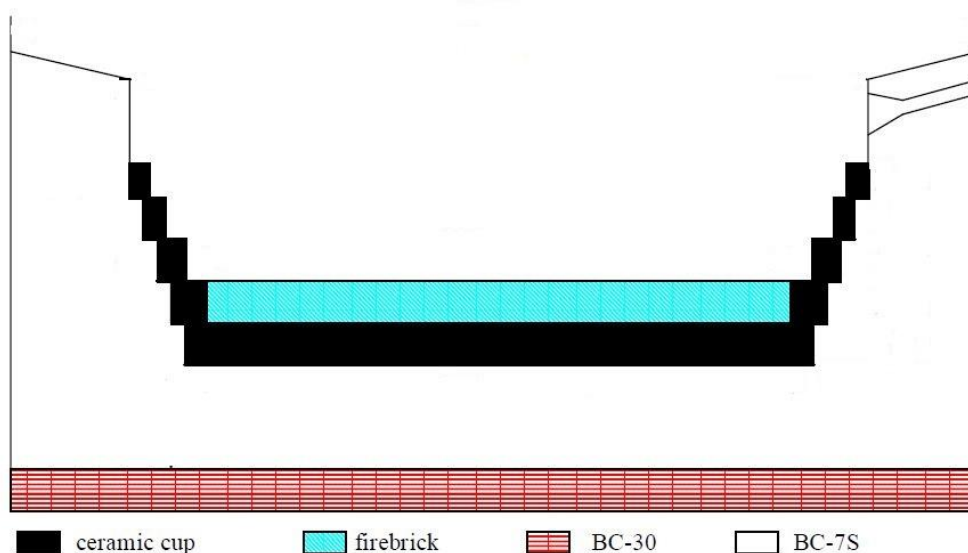
FIG 3: *Simple blast furnace hearth geometry*

1.4.2. Working of hearth:

Though most of the coke charged into the blast furnace burns at the tuyere level, a small fraction of it descends into the hearth where it dissolves in the metal to its near saturation. The entire charge is molten and it tends to stratify into slag and metal layers in the hearth from where these are tapped separately. These liquids are dense without pores and voids leading to the reduction in volume. So the cross section of the furnace below the tuyere level decreases. The hearth is the smallest cross section of the furnace [1].

1.4.3. Importance of modelling of hearth:

Due to the high temperatures of slag and molten iron, the refractory fireclay brick lining at the base of the hearth gets eroded. Wear of these refractories governs the campaign life of a blast furnace. With the erosion of the fireclay brick, the life span of a blast furnace is decreased. The vigorous operating conditions and the high temperature make it practically impossible for direct measurement and visualisation of processes within the hearth. Hence physical and mathematical models play an important role in understanding and assessing the cause-effect phenomena between the liquid iron, coke-bed and refractories.

CHAPTER 2

LITERATURE REVIEW

For many years, Nippon Steel Corp., Japan has been developing mathematical models of blast furncaces. Shinroku MATSUZAKI et al. [4] used such a model to analyze the conditions pertaining to blast furnace operations, formulate blast furnace operation policies, stabilize the blast furnace operations and lower the reducing agent ratio. They predicted that in the future, it was necessary to integrate a multitude of models and to develop a total blast furnace visualisation system. The future tasks also included the development of non-steady state models, three dimensional models and discrete models of solid particle behaviour.

P K Iwamasa *et al.* [3] used a commercial fluid dynamic package called TASCflow to simulate the iron flow and the heat transfer and thereby to estimate the refractory erosion in the hearth region of a blast furnace. The Flat Products Division Port Kembla Steelworks Blast Furnace 6 was studied in this case. A parametric survey of various operating conditions for PK6 was done and the production rates, coke bed porosity, dead-man shape, iron level and inlet temperatures were predetermined. Then several simulations were run by changing one parameter and keeping others constant. The results suggested that the shape and the location of the dead-man coke influence the wear of hearth refractory the most. The refractory wear is most rapid at the time when the coke is floating in the bottom of the hearth. Due to the vigorous recirculation zone driven by natural convection, the greatest erosion of refractory lining occurs at the walls and at the bottom of the hearth near the walls.

Vladimir Panjkovic *et al.* [5] have studied the temperature distribution profile and the wear of refractories in the hearth portion of BHP's Port Kembla No. 5 Blast Furnace (PK5BF). They used CFX 4.2 commercial package in their study and successfully developed a computational fluid dynamics model of iron flow and heat transfer in the hearth. The model was evaluated by placing four sidewall thermocouples and twenty pad thermocouples. After 1995, there was a marked increase in the temperatures at the pad thermocouples which suggested the erosion of the fireclay brick and the sidewall refractories. The model under-predicts the temperature near the centre of the hearth pad as well as the gradient in the pad area closer to the walls. The discrepancy between the measured and the calculated temperatures was attributed to a) larger floating dead-man porosity, b) greater erosion of refractories, c) unknown thermal conductivity of refractories under actual conditions and d) the spatial variation of temperatures on the cold face of refractories. The porous and non-porous volumes, large scales (diameter of hearth is greater than 10 meters), natural and forced convection and a wide range of velocities (from several m/s to less than 1 nm/s) presented interesting challenges to the CFD modelling [6].

Bryan WRIGHT et al. [6] refined the model of BlueScope Steel's No. 5 Blast Furnace hearth which was earlier used by Vladimir Panjkovic et al. [5]. This model developed by BlueScope Steel is known as Coupled Flow-Refractory Model (CFRM) which has been continuously improved over the years [7]. Bryan WRIGHT et al. included the ability to model different shaped coke free layers and a more accurate description of boundary conditions of the blast furnace hearth in the new model. It was prepared by using the commercial package CFX 4.4. The PK5BF furnace was under observation from 1st to 15th of April, 2002. The study focussed on the hearth refractory condition rather than on the short term variation in the thermocouple temperatures. The refractory configuration was predicted by adopting a trial and error approach, where refractory was gradually removed until a reasonable match was obtained between calculated and measured temperatures. The refractory assessment was then further refined through investigation of the effect of various coke free layer.

D. MALDONADO et al. [7] separately investigated the reason behind the erosion of hearth of Port Kembla No. 5 Blast Furnace by doing numerous simulations with the help of CFX 4.4 fluid flow package. Pad and sidewall thermocouples were located in the refractory. They considered the time period of January 2002 to January 2006 for their case study. The temperature of centre pad was 510° C during March 2002. It was simulated by using a uniform 400 mm coke free layer. There was a gradual reduction of refractory temperature during March 2002 to October 2003. It was proposed that low-conducting build-up or deposit layers were gradually formed during this period of time. Then there was a rapid increase in the temperature peaking late December 2003 (increasing to March 2002 level) which was attributed to the formation of a floating coke bed with a raised hemispherical shaped coke free layer. This was followed by another gradual decrease in the temperature through to March 2004. Since then the temperature fluctuated between 400-500° C.

Bao-Yu GUO et al. [8] modified the three dimensional computational fluid dynamics model of Port Kembla No. 5 Blast Furnace which was used for liquid iron flow and coupled refractory-liquid heat transfer by Vladimir Panjkovic et al. [5] with improved computational mesh quality and mathematical formulation. An extra thermal dispersion term and high order turbulence model were included in the new model to treat the coke free zone and the coke bed region consistently. The simulation results indicated that:

i) Natural convection is very significant in controlling the global flow pattern of the iron melt. Due to the significant effect of the thermally-driven buoyancy flows, the sensitivity

of calculated flow to the coke free layer was reduced. This phenomenon further helped in clarifying the effect of coke free layer.

ii) A well organised iron flow patterns was revealed from the study. The experiment included six coke bed configurations. Two distinct regions of iron flow were formed from the top to the bottom for all the configurations. A stationary flow zone existed in the bottom half of the coke bed above the coke free layer. The temperature of iron varied mostly in vertical direction.

iii) The temperature distribution of pad gradually became less dependent on the inner conditions of the hearth. Instead, it was controlled by the heat transfer across the hearth refractory. Hence the determination of the refractory properties and the surface profile under the observed operating state is critical for the reliable and accurate prediction of the flow and heat transfer in the hearth.

This model was prepared by using ANSYS-CFX fluid flow package and was limited to a steady-state, single phase flow condition.

W. T. Cheng et al. [9] tried to examine the shear stress on the wall of Port Kembla No. 5 Blast Furnace hearth by solving the turbulent Navier-Stokes equation and thermal-energy-balance equation. The computation was performed with the use of FLUENT package (version 6.1). The results of the study revealed that:

i) The peak value of shear stress was decreased due to an increase in the tap hole length.

ii) The location of the peak value of shear stress coincides with the location of higher temperature actually measured on the hearth wall. It signifies enhanced heat transfer to the wall at location of peak stress.

iii) The shear stress on the furnace wall is reduced at three axial planes due to the presence of the tap hole blocks. These planes are namely; below the tap hole plane, at the top hole exit plane and above the tap hole plane.

Chen-En HUANG et al. [10] prepared a case study on the hot metal behaviour in the hearth of No. 2 blast furnace at China Steel Corporation by developing a numerical model with the use of computational fluid dynamics. Their model solved the three dimensional Navier-Stokes equations with conjugate heat transfer and Darcy's law for hot metal flowing through the dead-man with porous structure. The summary of the observations:

i) When the dead-man becomes sitting with a gutter coke-free space, the circulatory hot metal is increased. As a result, the temperature at the hearth corner increases. This

suggests that the existence of gutter coke-free space may cause elephant foot type erosion.

ii) With the floating dead-man operation, more hot metal tends to travel to the hearth bottom before it reaches the tap hole. This flow pattern may enhance the renewal of coke by dissolving carbon into the hot metal and may avoid the dead-man from clogging.

iii) During the casting operation, the temperature and heat flux in the tap hole region significantly increases. Individually monitoring the temperature change of the cooling water of the copper staves and installing thermocouples around the tap holes may be needed for the sake of safety.

iv) The conduction of the linings and the cooling elements dominate the heat transfer resistance of the hearth. Therefore, the cooling efficiency cannot be improved by reducing the temperature of the cooling water. This implies that the use of the water chillers in the beginning of the blast furnace campaign may not be able to have a significant effect on the protection of the hearth.

Shigeru UEDA *et al.* [11], in their research paper, showed the recent progress and future prospective on mathematical modelling of blast furnace. Currently low reducing agent operation has been actively pursued. The inner volume of the blast furnace has been frequently enlarged over 5000 m^3 at the relining of blast furnace in Japan to secure the increase in crude steel production rate. From the above reasons, the precise and reliable control of large blast furnace based on the low reducing operation is required to attain the above target smoothly. In the past 50-55 years, mathematical models of ironmaking processes have been developed applying mainly CFD modelling. The operational efficiency of the processes has been significantly improved by the derived models. The development of advanced mathematical models of blast furnace such as Discrete Element Method (DEM) has attracted a special attention in ironmaking field to satisfy the above requirements. The combined model of DEM with the continuum model is under development for the purpose of the accurate understanding of the inner state of blast furnace. The mathematical process models should be studied much more together with optimization and global evaluation for effective production and substantial decrease in environmental load.

Jim R. POST *et al.* [12] made two dimensional CFD calculations with simple geometry of blast furnace hearth. They studied the effects on the density changes of the hot metal due to cooling and carbon dissolution. They also showed the effect of buoyancy on hot metal flow and the effect on carbon dissolution rate due to fluid speed. Refractory wear was evaluated by

analysing fluid flow along the wall since carbon dissolution from refractory bricks is similar compared to carbon dissolution from cokes. 2D simulations resulted in:

i) Stratification of hearth into two major fluid speed zones. They are a) high fluid speed zone and b) low fluid speed zone.

ii) Division of lower zone into a) high carbon concentration region and b) low carbon concentration region due to hot metal circulation in the low fluid speed zone.

iii) High carbon dissolution rates below and around the tap hole and straight across at the opposite wall.

Although the general furnace shape and the tap hole shape model did not resemble a real blast furnace hearth, the significance of the competing effects for a real life 3D situation could be seen from the 2D simulation results.

Steven Vernengo *et al.* [13] used a comprehensive CFD model on a simplified structure of dead-man to simulate a large commercial scale blast furnace hearth. The erosion profile, the liquid iron flow pattern and temperature distribution in the liquid and the refractories in the hearth were analyzed along with a limited parametric study.
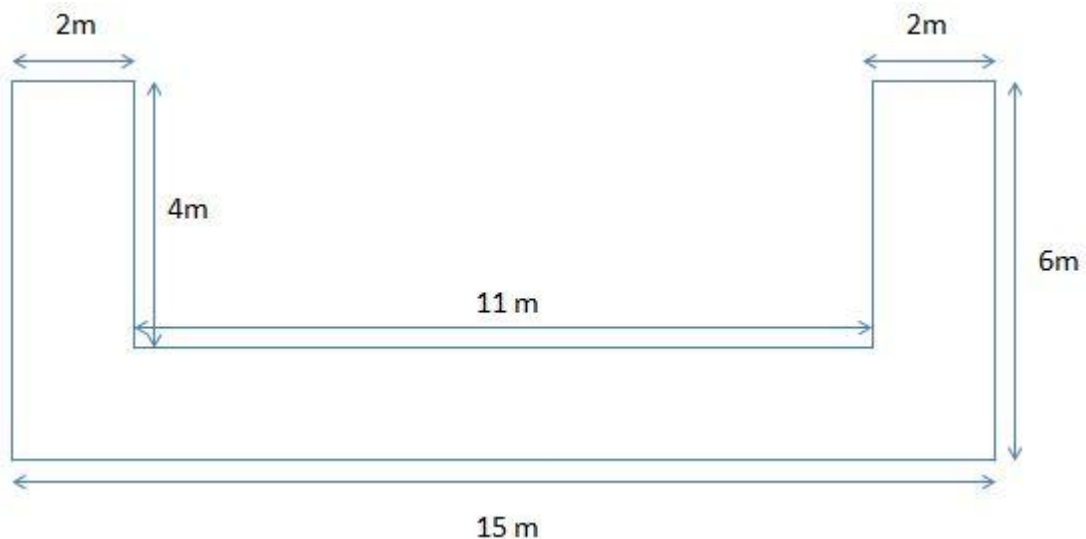
CHAPTER 3

SIMULATION DETAILS

3.1. Modelling of hearth:

Two dimensional models of a simple blast furnace hearth have been developed using the commercial CFD package ANSYS 15.0 Workbench. The following figure shows the schematic diagram of the hearth.

FIG 4: *Schematic diagram of the hearth used in the study*

3.1.1. 2D modelling of only hearth using Fluid Flow (FLUENT):

3.1.1.1. Geometry:

1) Select "Fluid Flow (FLUENT)" under "Analysis Systems".
2) Click on "Geometry".
3) Change the "Analysis type" to 2D under "Advanced Geometry" option in the right hand side corner.
4) Double click on "Geometry". Another window will be opened.
5) Select "Meter" for defining geometry dimensions.
6) Click on "Sketching", then on "Settings".
7) Select "Grid". Turn on 2d and Snap option.
8) Select "Poly line" under "Draw".
9) Draw the diagram as per the given dimensions.
10) Click on "Coincide" under "Constraints" command and fix one side of the hearth with the Y-axis.
11) Assign proper dimensions to the figure under "Dimensions" command.

12) Go to "Concept". Select "Surfaces from Sketches".

13) Click on the base line of the geometry. Then click on "Apply" in left hand side bottom corner.

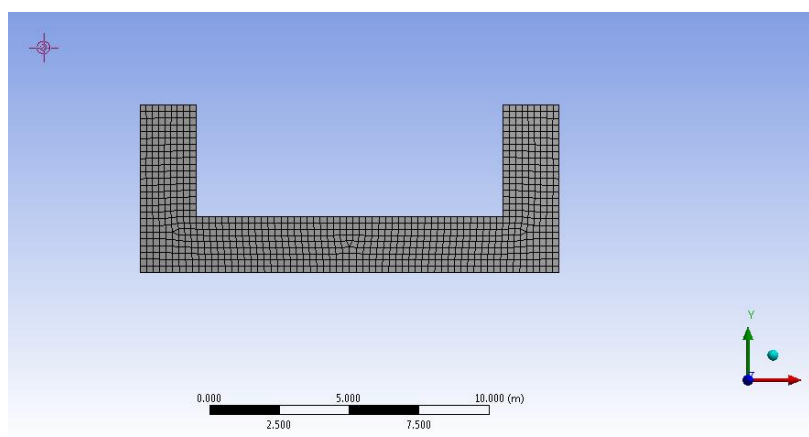14) Then "Generate" the diagram.

The diagram will look like this:
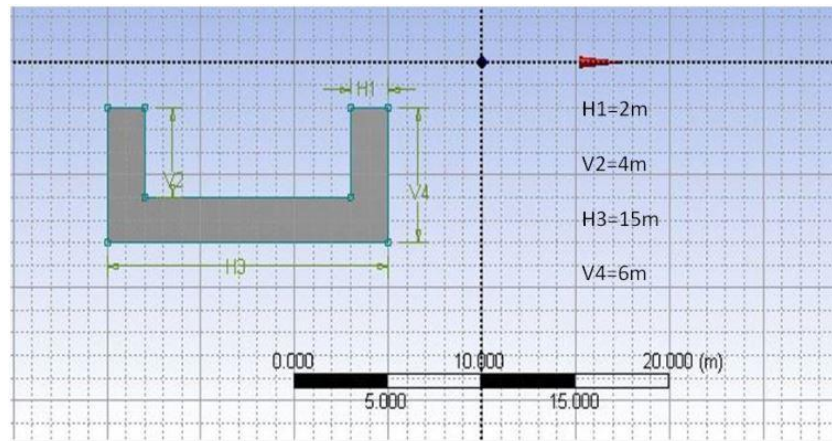
FIG 5: *Hearth model*

15) Close the window and "Update Project".

3.1.1.2. Meshing:

1) Double click on "Mesh". A new window will be opened.

2) Click on "Mesh".

3) Select "Fine" "Relevance Centre" under "Sizing".

4) Then "Generate Mesh".

FIG 6: *Meshing of the model*

5) Select "Edge". Select the upper part of hearth base and right click on that part and select "Create Named Section" under that. Write "Inner Wall". Then follow the same process for outer wall of the hearth.

The "Inner Wall" and "Outer Wall" will look like this:

FIG 7 (a): *Inner Wall*   **FIG 7 (b):** *Outer Wall*

6) Close the window and "Update Project".

3.1.1.3. Setup:

1) Double click on "Setup". Select "Double Precision" option. A new window will be opened showing a diagram like this:
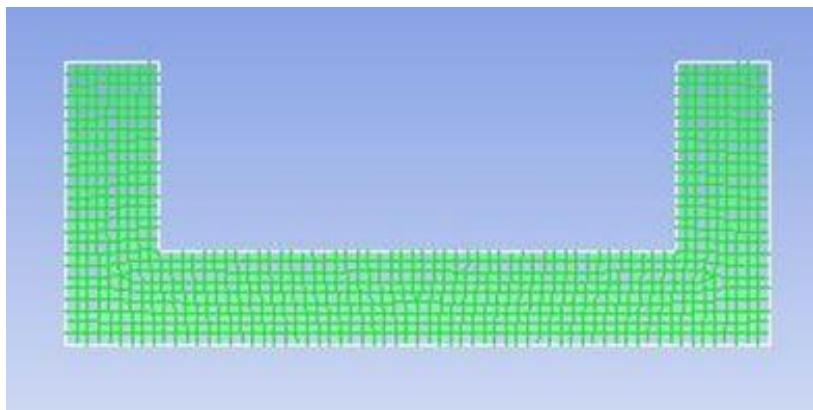
FIG 8: *Initial setup diagram*

2) Go to "Models" under "Solution Setup". Turn on the "Energy" equation.
3) Go to "Materials", select "Solid". Click on "Create/Edit". Create new carbon material and assign the density, specific heat and thermal conductivity values of carbon.
4) Select the type of the "surface body" under "Cell Zone Conditions" as "Solid". Edit the "Surface body" to "Carbon".

5) Go to "Boundary Condition". Click on "Inner Wall". "Edit" it. Select "Thermal". Set the temperature at 1700 K. The same process is repeated for assigning boundary condition for "Outer wall". Outer wall is at room temperature i.e. 300 K.

6) Then "Edit" the "Residual, Statistics and Force" under "Monitors". Change the energy value to $1e^{-15}$.

7) Select "Solution Initialisation" under "Solution" option. Select Initialisation Methods as "Standard Initialisation". Compute from "Inner Wall" and then "Initialise".

8) Select "Run Calculation" under "Solution". Take 50 numbers of iterations. Click on "Calculate".

9) Go to "Graphs and Animation" under "Results". Click on "Contours" under "Graphics". Select contours of "Temperature". Select "Filled" under "Option". Click on "Display". The result of the temperature distribution profile will be displayed.

3.1.2. 2D modelling of hearth containing only hot metal by using Fluid Flow (FLUENT):

3.1.2.1. Geometry:

The hearth geometry was prepared in the same way as in the previous case. This time the geometry of hearth was designed only to model the hot metal part. The following figure shows the hearth diagram:
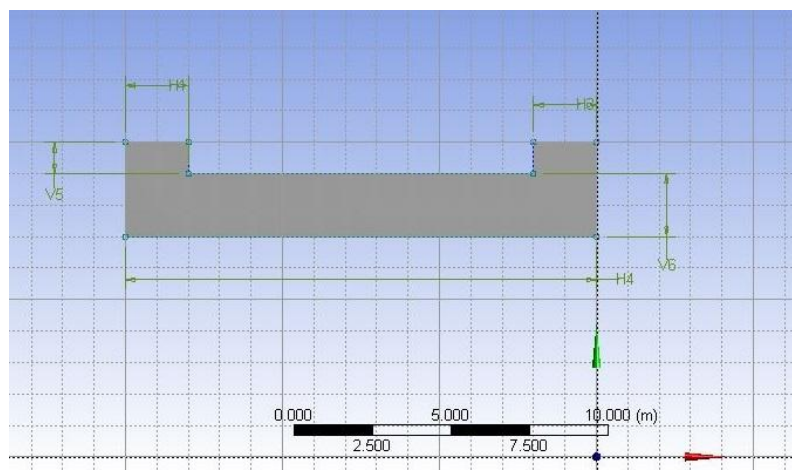

FIG 9: *Hearth model*

The following steps state how to generate fluid (hot metal) inside the hearth.

1) Click on "New Sketch". Select "Sketch2".

2) Now select "Poly line" under "Draw" and plot a rectangular part. "Coincide" its edges with the inner walls of the hearth under "Constraints" command.

3) Then "Generate" "Surface from Sketches".

4) Change the "Operation" to "Add Frozen" for both the surface sketches.

5) Select both "Surface Body" under "2 Parts, 2 Bodies" and "Form New Part".

6) Change the details of the newly formed part by assigning proper "Solid" and "Fluid" type for hearth and molten metal, respectively.
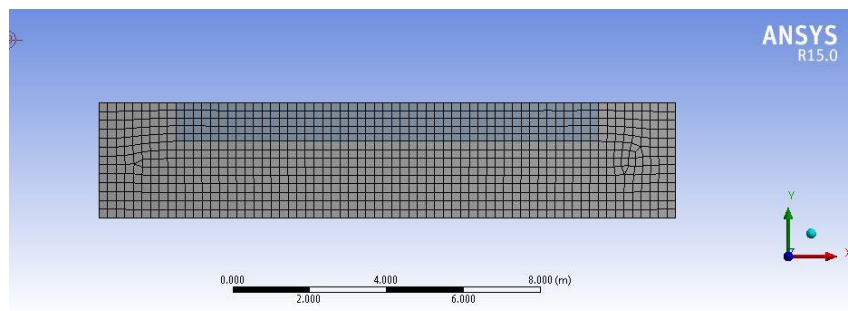
The 2 parts model is shown below:
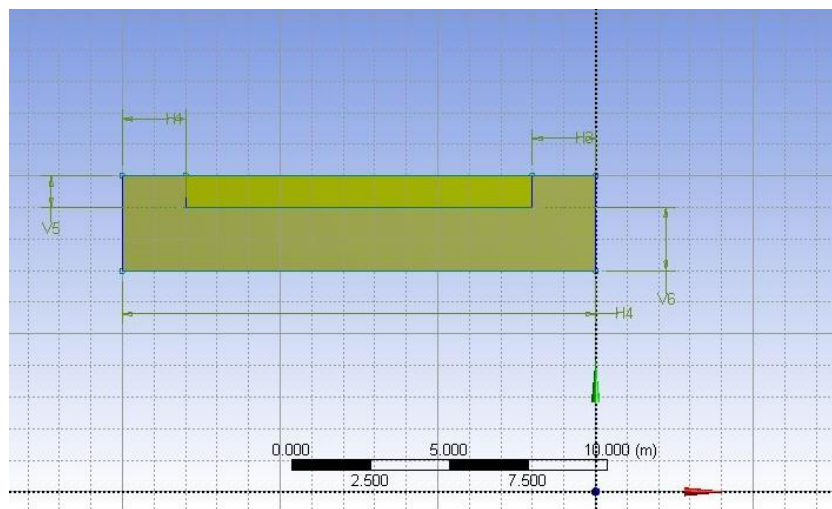
FIG 10: *Model of hearth and hot metal*

3.1.2.2. Meshing:

Same steps are followed for meshing as in the previous case. Mesh size can be made finer as per requirement by changing the relevance centre. The following picture shows the hearth body after meshing.

FIG 11: *Meshing of two part body*

Here the only change is in assigning the "Named Selection". The upper part of the hearth metal is named as "Input". Then follow the same process for naming the outer wall of the hearth.

The "Input" and "Outer Wall" will look like this:
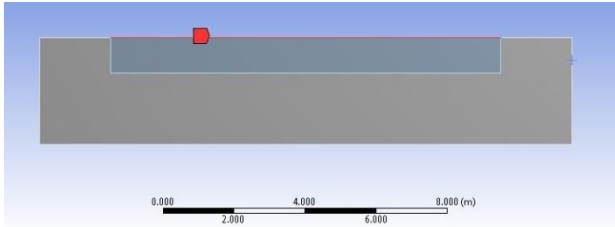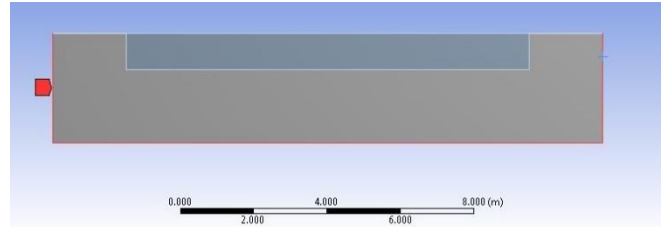
| **FIG 12 (a):** *Input* | **FIG 12 (b):** *Outer Wall* |

Select "Update Project" after closing the "Mesh" window.

3.1.2.3. Setup:

1) Double click on "Setup". Select "Double Precision" option. A new window will be opened showing a diagram like this:
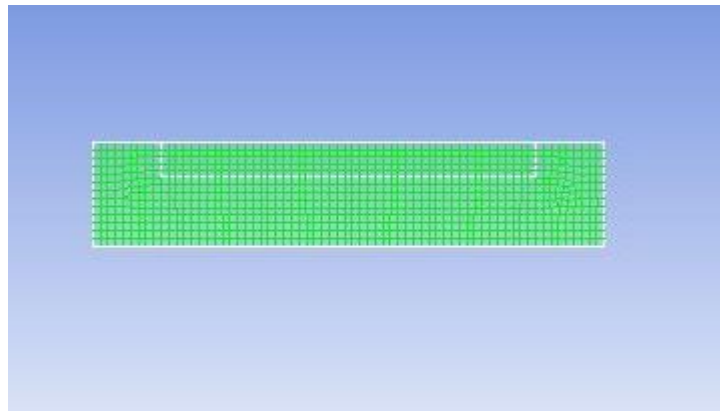
FIG 13: *Initial Setup Diagram*

2) Go to "Models" under "Solution Setup". Turn the "Energy" equation on.

3) Go to "Materials", select "Fluid". Create new fluid material ("Molten Iron" in this case) by editing it and assign its physical properties such as density, specific heat and thermal conductivity. Repeat the procedure for "Solid" as done in the previous case.

4) Go to "Cell Zone Conditions". Select "fluid-part-surface body". Set the type as "Fluid". Change the material name to "Molten Iron". Then select "part-surface body". Here type is "Solid". "Edit" it to change the material name to "Carbon".

5) Then go to "Boundary Condition". Click on "Input", then on "Edit". Select "Thermal". Set the temperature at 1800 K. The same process is repeated for assigning boundary condition for "Outer wall". Outer wall is at room temperature i.e. 300 K.

6) Then "Edit" the "Residual, Statistics and Force" under "Monitors". Change the energy value to $1e^{-15}$.

7) Select "Solution Initialisation" under "Solution" option. Select Initialisation Methods as "Standard Initialisation". Compute from "Inner Wall" and then "Initialise".

8) "Run Calculation" under "Solution". Take 50 iterations. Click on "Calculate".

9) Go to "Graphs and Animation" under "Results". Click on "Contours" under "Graphics". Select contours of "Temperature". Select "Filled" under "Option". Click on "Display". The result of the temperature distribution profile will be displayed.

3.1.3. 2D modelling of hearth containing both hot metal and molten slag by using Steady-State Thermal:

3.1.3.1. Geometry:

1) Select "Steady-State Thermal" under "Analysis Systems"

2) Draw the 3 body, 3 part diagram by following the steps of second case and "Generate" the diagrams by applying "Surfaces from Sketches".

3) Change the "Operation" to "Add Frozen" for all the surface sketches.

4) Select 3 "Surface body" under "3 Parts, 3 Bodies" and "Form New Part".

5) Change the "Details" of the newly formed part by assigning proper "Solid" and "Fluid" types for hearth, slag and molten metal.

   The following figure shows the 3 part body of hearth.
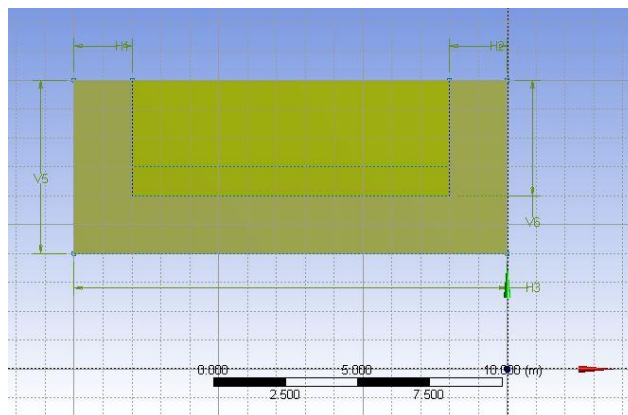
**FIG 14:** *Model of hearth, hot metal and slag*

3.1.3.2. Engineering Data:

1) Double click on "Engineering Data".
2) Add new "Carbon" material. Click on "Filter Engineering Data". Select "Density" from "Physical Properties" and assign the density value of carbon. Select "Isotropic Thermal Conductivity" from "Thermal" and assign the corresponding value. Also give the value of "Specific Heat" of carbon.
3) Add new hot metal and slag and follow the same procedure to assign the values of various physical quantities for respective materials.
4) Close the "Engineering Data" and "Update Project".

3.1.3.3. Modelling:

1) Double click on "Model".
2) Select hearth surface body under "Part" of "Geometry". Assign the thickness of hearth and select "Carbon" "Material" as the "Assignment".
3) Then follow the same step to assign the thicknesses of slag and hot metal.
4) Click on "Mesh". Make the "Relevance Centre" "Fine" and "Generate Mesh".
    The following figure shows the model obtained after meshing.
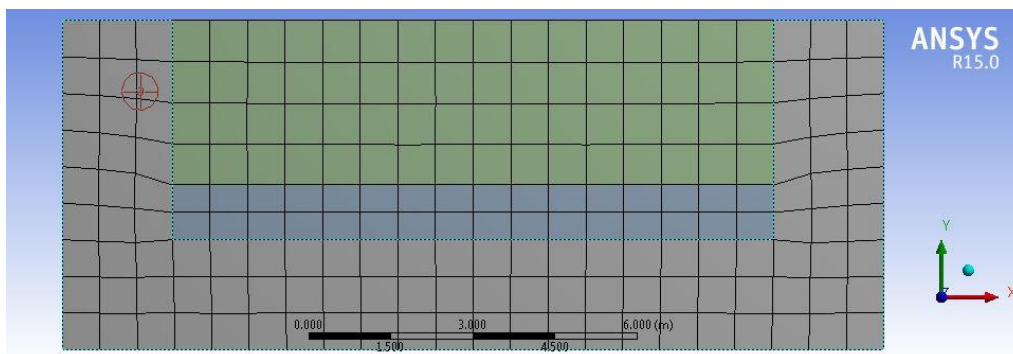
FIG 15: *Meshing of three part body*

5) Select "Steady-State Thermal" and "Insert" Temperature". Click on "Edge" and then select the upper part of the slag surface. Then "Apply" at the "Geometry". Give the temperature "Magnitude" 1600° C.
6) Follow the same procedure to add one more temperature condition to give the temperature "magnitude" at the "Outer Wall" 27° C.
7) Change "Number of Steps" to 50 at "Analysis Settings" under "Steady-State Thermal".

8) Then "Insert" "Temperature" under "Thermal for "Solution". Then "Solve".

3.2. Material Properties:

The following table shows the standard values of model parameters [5].

	Density (Kg/m^3)	Specific Heat (J/Kg-K)	Thermal Conductivity (W/m-K)
Carbon	2267	7000	2705
Hot Metal (Iron)	1260	850	-
Slag	15.5	16.5	5.4

Table 1: *Physical properties of the material used in the simulation*

3.3. Governing Equations:

In our simulations, we have used only the conduction equation. Conduction refers to the heat transfer between two bodies through molecules which are, more or less, static [14].

Fourier's law of heat conduction states that the rate heat transfer is linearly proportional to the temperature gradient.
Mathematically,

$$q \ \alpha \ \frac{dT}{dx}$$

(Or)

$$q = -k\frac{dT}{dx} \qquad (3.1)$$

where, q is the rate of heat flux in W/m^2, dT/dx is the temperature gradient in the direction of heat flow x and k is the proportionality constant. It is a property of the material thorugh which heat propagates. This property of the material is called *thermal conductivity* (W/m-K). The negative sign in the temperature gradient shows that the heat flows from a high temperature to a low temperature. So, the slope dT/dx is negative. Since we consider temperature difference in the temperature gradient term, we can take the temperature either in ^{O}C or in Kelvin (1^O C = 1 K).

Let us consider the following case.
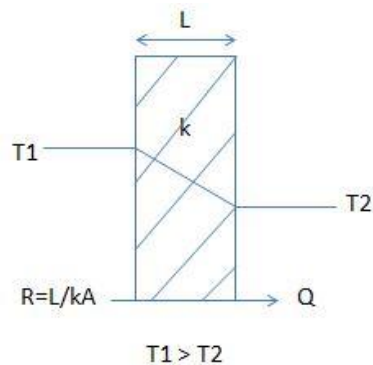
FIG 16: *Thermal resistance offered by a wall of thickness L*

For the simple case of steady-state one-dimensional heat flow through a plane wall of thickness L, the temperature gradient and the heat flow do not vary with time. So, by integrating the equation (3.1) with its proper limits, we can write:

$$q \int_0^L dx = - \int_{T1}^{T2} k dT$$

where, the temperature at the left surface (at x=0) is uniform at T1 and the temperature at the right face (at x=L) is uniform at T2.

Since k is independent of T, after integration, we obtain:

$$q = k \left(\frac{T1 - T2}{L} \right) \tag{3.2}$$

If A is the cross sectional area of the surface normal to the heat flow, then the rate of heat transfer is given by:

$$Q = qA = kA \left(\frac{T1 - T2}{L} \right)$$

If cross sectional area is unity, then the above equation becomes:

$$Q = k \left(\frac{T1 - T2}{L} \right) \tag{3.3}$$

The above equation can also be written as:

$$Q = \frac{T1 - T2}{R}$$

where, R=L/k (for unit cross sectional area) is known as the conductive thermal resistance to heat flow offered by the wall.

Now, let us consider the case of a composite wall of *unit cross sectional area*.
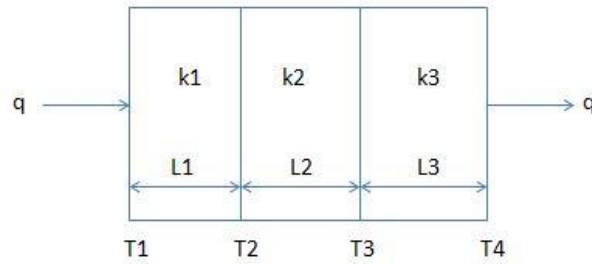
FIG 17: *Conduction through three resistances in series*

For a composite wall, as shown in FIG 16, there are three resistances in series. The rate of heat conduction is the same through all sections. The total thermal resistance is given by:

$$R = R1 + R2 + R3 = (L1 / k1) + (L2 / k2) + (L3 / k3)$$

The rate of heat conduction is given by:

$$Q = \frac{T1-T4}{R} = \frac{T1-T4}{\frac{L1}{k1} + \frac{L2}{k2} + \frac{L3}{k3}} \tag{3.4}$$

Also,

$$Q = (T1 - T2) * (k1 / L1) = (T2 - T3) * (k2 / L2) = (T3 - T4) * (k3 / L3) \tag{3.5}$$

CHAPTER 4

SIMULATION RESULTS

&

DISCUSSIONS

4.1. Simulation of only hearth:

4.1.1. Assumptions:

The following assumptions were made during the simulation:

i) The process is a steady-state process.

ii) The presence of hot metal and slag is neglected.

iii) The presence of refractory fireclay brick is not taken into account.

iv) Chemical reactions due to slag and hot metal are neglected.

4.1.2. Boundary conditions:

The following boundary conditions are imposed.

i) The temperature of the inner side of the hearth is taken to be 1450°C (1700 K).

ii) The temperature of the outer walls of the hearth is taken to be 27°C (300 K).

iii) The top surface of the refractory wall is adiabatic.

iv) The process is a two dimensional heat transfer process.

v) Cross sectional area is taken to be unity.

4.1.3. Temperature profile:

The following figure shows the number of iterations taken to complete the simulation process.

FIG 18: *Convergence of the simulation*

The temperature profile obtained after the simulation process is shown below.

| 1.70e+03 |
| 1.63e+03 |
| 1.56e+03 |
| 1.49e+03 |
| 1.42e+03 |
| 1.35e+03 |
| 1.28e+03 |
| 1.21e+03 |
| 1.14e+03 |
| 1.07e+03 |
| 1.00e+03 |
| 9.30e+02 |
| 8.60e+02 |
| 7.90e+02 |
| 7.20e+02 |
| 6.50e+02 |
| 5.80e+02 |
| 5.10e+02 |
| 4.40e+02 |
| 3.70e+02 |
| 3.00e+02 |

FIG 19: *Temperature distribution profile across the section of the hearth*

4.2. Simulation of hearth containing hot metal:

4.2.1. Assumptions:

The following assumptions were made during the simulation:

i) The process is a steady-state process.

ii) The presence of slag is neglected.

iii) The presence of refractory fireclay brick is not taken into account.

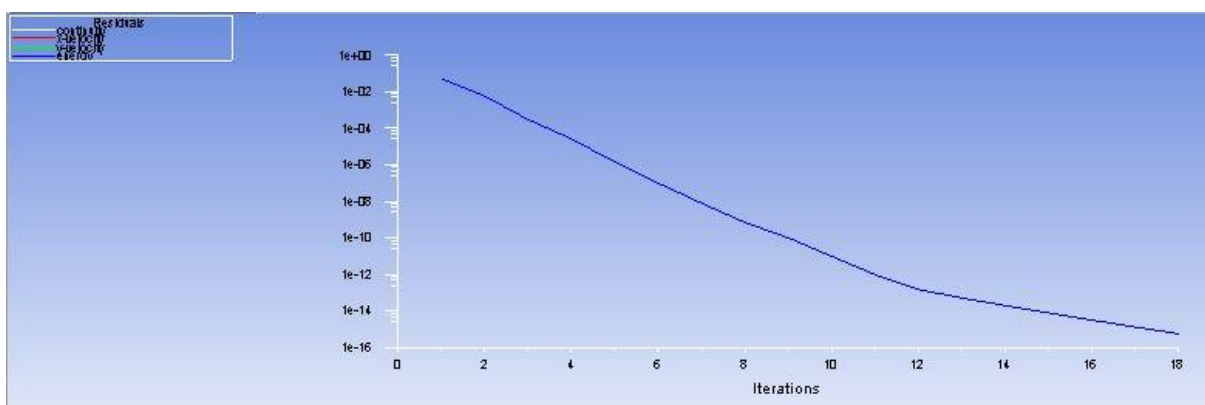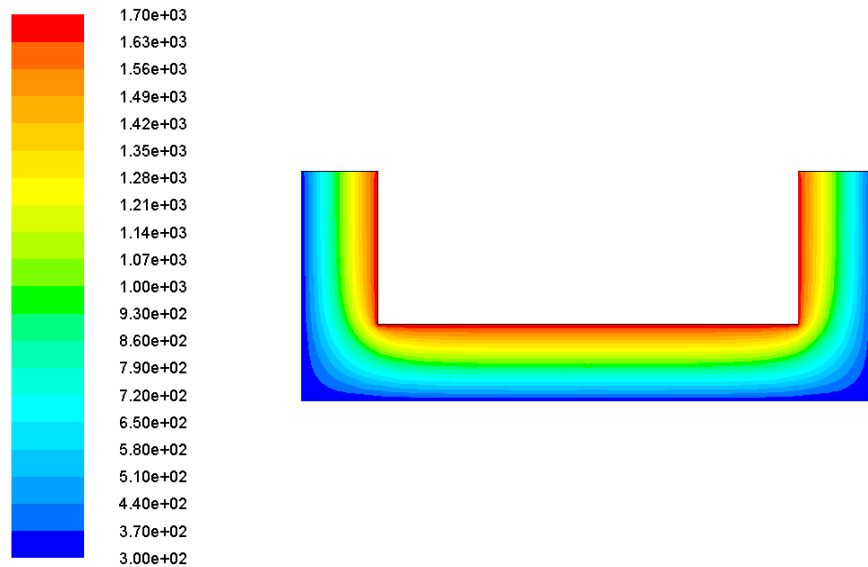iv) Chemical reactions between hearth and hot metal are neglected.

v) The free surface of liquid iron is flat and horizontal.

vi) Solidification is neglected.

vii) Heat transfer takes place by conduction.

viii) There is no mass transfer between the liquid iron and refractory walls.

4.2.2. Boundary conditions:

The following boundary conditions are imposed.

i) The liquid iron level is constant.

ii) The top surface of the refractory wall is adiabatic.

iii) The temperature of the outer surface of the molten hot metal is taken to be 1550° C (1800 K).

iv) The outer walls of the hearth are at room temperatures i.e. at 27° C (300 K).

v) The process is a two dimensional heat transfer process.

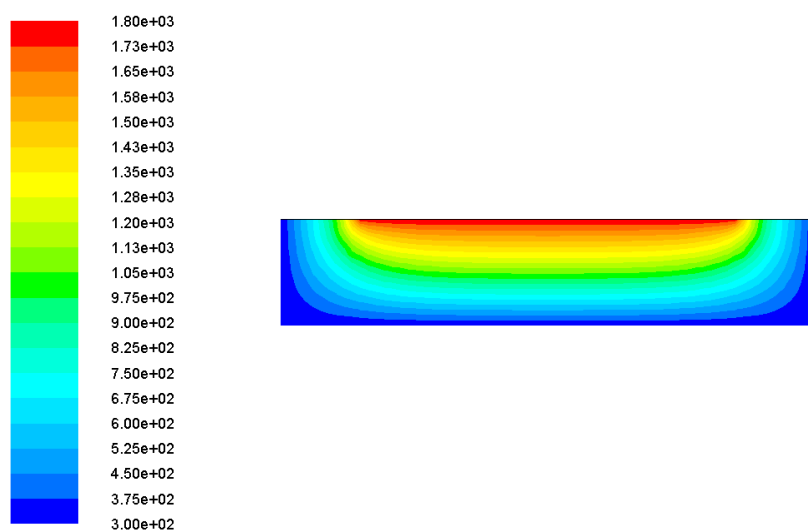vi) Cross sectional area is taken to be unity.

4.2.3. Temperature profile:

The following figure shows the number of iterations taken to complete the simulation process.
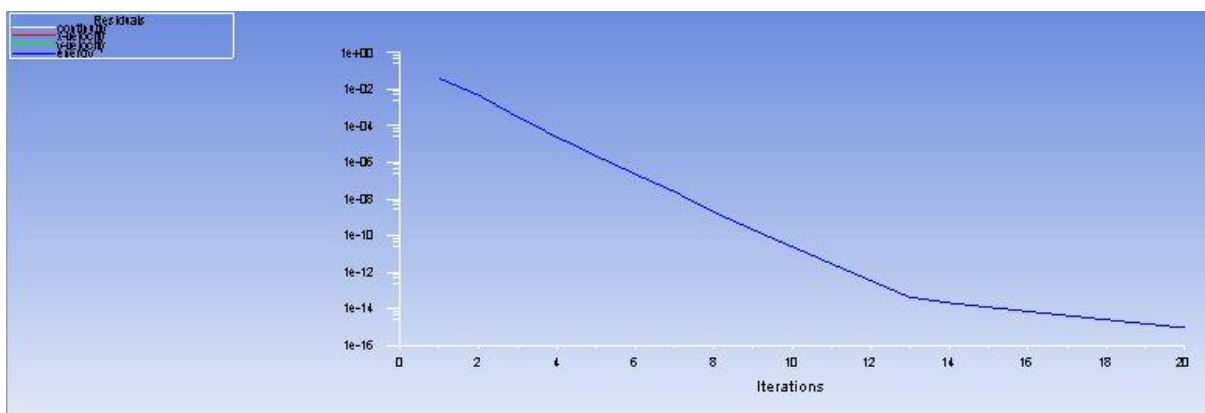
FIG 20: *Convergence of the simulation*

The temperature profile obtained after the simulation process is shown below.

FIG 21: *Temperature distribution profile across the section of the hearth*

From the temperature distribution contour, it is observed that the temperature at horizontal interface of the hearth and hot metal is around 1300 K. By solving the governing equation (3.5), we obtain the temperature value to be 371 K. This large discrepancy between the observed and calculated temperature value may be due to too much of assumptions.

4.3. Simulation of hearth containing both hot metal and molten slag:

4.3.1. Assumptions:

The following assumptions were made during the simulation:
i) The process is a steady-state process.
ii) The presence of refractory fireclay brick is neglected.
iii) Chemical reactions between hearth, hot metal and slag are not considered.
iv) The free surface of molten slag is flat and horizontal.
v) Solidification is neglected.
vi) Heat transfer takes place by conduction.
vii) There is no mass transfer between the slag, liquid iron and refractory walls.
viii) There is no mixing of liquid metal and slag.

4.3.2. Boundary conditions:

The following boundary conditions are imposed.
i) The temperature of the outer surface of the molten slag is taken to be 1600° C (1900 K).
ii) The outer walls of the hearth are at room temperatures i.e. at 27° C (300 K).
iii) The liquid iron level is constant.
iv) The top surface of the refractory wall is adiabatic.
v) The process is a two dimensional heat transfer process.
vi) Cross sectional area is taken to be unity.
vii) Density of slag is taken for 16.3% FeO, 2.06% Fe_2O_3, 33.4% CaO, 10% MnO and 38.2% Si contents.

### 4.3.3. Modelling by using Steady-State Thermal:

In case of more than two solid-solid, solid-liquid or liquid-liquid systems, it is impossible to differentiate between materials those are in same phase using Fluid Flow (FLUENT) which creates problem at the time of giving boundary conditions. Moreover, since we are using only conduction method for heat transfer, it is advantageous to use Steady-State

Thermal instead of Fluid Flow (FLUENT). We don't need to assign density, viscosity or any other fluid material properties for simple heat transfer problem. We can add as many parts as we want to a given system without facing any difficulty while assigning boundary conditions.

4.3.4. Temperature profile:

The following figure shows the temperature profile obtained after the simulation process.
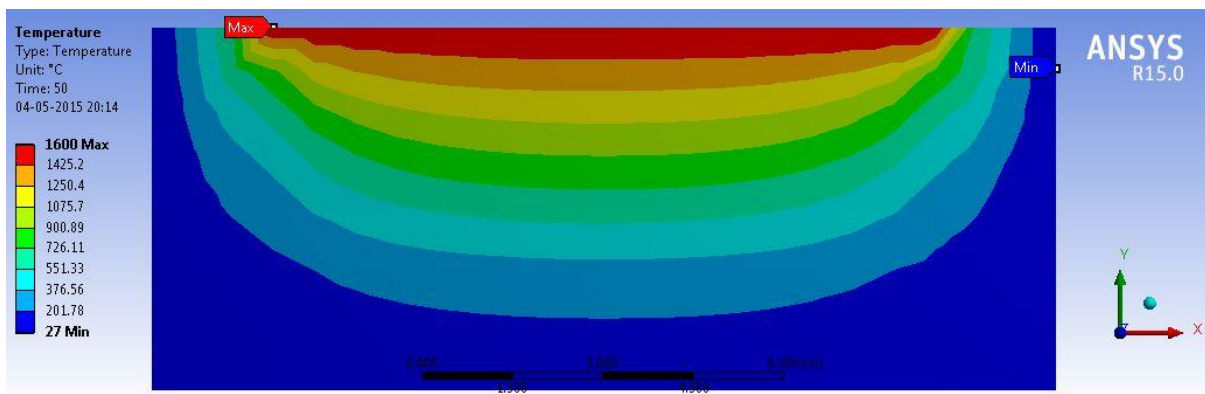
FIG 22: *Temperature distribution profile across the section of the hearth*

From the temperature distribution contour, it is observed that the temperature at horizontal interface of the hearth and hot metal is around 640° C. By solving the governing equation (3.4) and (3.5), we obtain the temperature value to be around 500° C. Again, the temperature at the horizontal interface of hot metal and molten slag is observed to be 900° C. From calculation, the temperature is found out to be around 1500° C. This large discrepancy between the observed and calculated temperature value may be due to too much of assumptions which lead to imprecision in simulation conditions.

CHAPTER 5

CONCLUSION

A simple blast furnace hearth is modelled by using ANSYS 15.0 Workbench. Several simulations are run with the help of Fluid Flow (FLUENT) and Steady-State Thermal. Only hearth, hearth with hot metal and hearth with hot metal and molten slag are the three cases which are considered for simulation program. Their temperature distribution profiles are displayed and studied. The discrepancy between the observed and the calculated data may arise from the use of too many assumptions in implementing the boundary conditions. We need to work on a few areas to overcome this problem.

Scope for future work:

A wide range of scope has been opened up for further improvement:

i) Fluid flow velocity contour can be obtained- we can have an idea about the impact of fluid velocity on the life of refractory materials.

ii) Effect of temperature on refractory wear can be analysed- we can get the temperature at the interface of the hearth and slag-metal system depending upon the physical properties of the material. As a result we can predict the life of refractory material.

iii) A three dimensional model can also be prepared- in two dimensional modelling, we have taken the interface of hearth-metal, metal-slag system as a conducting wall of negligible thickness. In 3D simulations, proper interfaces can be modelled from which we can have an idea how the interaction between the slag and metal may affect the temperature distribution profile.

REFERENCES

[1] Ahindra Ghosh, Amit Chatterjee, *IRONMAKING and STEELMAKING*

[2] Shan-Wen Du, Wei-Hsin Chen, John A. Lucas, "Pulverized coal burnout in blast furnace simulated by a drop tube furnace", *Energy 35 (2010) 576-581.*

[3] P K Iwamasa, G A Caffery, W D Warnica, S R Alias, "Modelling of Iron Flow, Heat Transfer and Refractory Wear in the Hearth of an Iron Blast Furnace", *Inter Conf on CFD in Mineral & Metal Processing and Power Generation, CSIRO 1997.*

[4] Shinroku MATSUZAKI, Akihiko SHINOTAKE, Masaaki NAITO, Tsunehisa NISHIMURA, Kazuya KUNITOMO, Takashi SUGIYAMA, "Development of Mathematical Model of Blast Furnace", *NIPPON STEEL TECHNICAL REPORT No. 94, JULY 2006*

[5] Vladimir PANJKOVIC and John TRUELOVE, "Computational Fluid Dynamics Modelling of Iron Flow and Heat Transfer in the Iron Blast Furnace Hearth", *Second International Conference on CFD in the Minerals and Process Industries, CSIRO, Melbourne, Australia, 6-8 December 1999*

[6] Bryan WRIGHT, Paul ZULLI, Frank BIERBRAUER and Vladimir Panjkovic, "Assessment of Refractory Condition in a Blast Furnace Hearth using Computational Fluid Dynamics", *Third International Conference on CFD in the Minerals and Process Industries, CSIRO, Melbourne, Australia, 10-12 December 2003*

[7] D. MALDONADO, P. ZULLI, B. Y. GUO and A. B. YU, "Mathematical Modelling of Flows and Temperature Distributions in the Blast Furnace Hearth", *Fifth International Conference on CFD in the Process Industries, CSIRO, Melbourne, Australia, 13-15 December 2006*

[8] Bao-Yu GUO, Daniel MALDONADO, Paul ZULLI and Ai-Bing YU, "CFD Modelling of Liquid Metal Flow and Heat Transfer in Blast Furnace Hearth", *ISIJ International, Vol. 48 (2008), No.12, pp. 1676-1685*

[9]   W. T. Cheng and C. N. Huang, "Simulation of Iron Flow and Heat Transfer in the Hearth of a Blast Furnace", *On-line Number 821*

[10]   Chen-En HUANG, Shan-Wen DU and Wen-Tung CHENG, "Numerical Investigation on Hot Metal Flow in Blast Furnace Hearth through CFD", *ISIJ International, Vol. 48 (2008), No. 9, pp. 1182-1187*

[11]   Shigeru UEDA, Shungo NATSUI, Hiroshi NOGAMI, Jun-ichiro YAGI and Tatsuro ARIYAMA, "Recent Progress and Future Perspective on Mathematical Modeling of Blast Furnace", *ISIJ International, Vol. 50 (2010), No. 7, pp. 914-923*

[12]   Jim R. POST, Tim PEETERS, Yongxiang YANG, Markus A. REUTER, " Hot metal Flow in the Blast Furnace Hearth: Thermal and Carbon Dissolution Effects on Buoyancy, Flow and Refractory Wear", *Third International Conference on CFD in the Minerals and Process Industries, CSIRO, Melbourne, Australia, 10-12 December 2003*

[13]   Steven Vernengo, Rade Milanovic, CHenn Q. Zhou, Pinakin Chaubal and D Huang, "Computations of Liquid Flow and Heat Transfer in the Hearth of a Blast Furnace", *ASME 2003 International Mechanical Engineering Congress and Exposition*

[14]   P K Nag, *Heat and Mass Ttransfer*