Thermal Analysis and Flow Visualization in Vacuum Furnace Using CFD

Gururaj Lalagi, Adarsha G C, Vedavyasa V, M S Rajagopal

Abstract

Vacuum furnace operates with high temperature gases which flow in multiple nozzles during cooling process. The pattern of gas flow inside hot chamber is observed. The present study investigates in detail, the effect of mass flow rate of coolant on hot chamber inside the vacuum furnace. Under this study, 2D and 3D CAD models with the row of 6 holes at equidistance from the center line considered. These models are tested to find the flow visualization and thermal analysis of coolant inside hot chamber using CFD analysis. After giving inputs values and calculating result by using FLUENT software, results are verified and further improvements are carried out to improve the performance and overall efficiency of vacuum furnace.

Keywords: Furnace, Vacuum pressure, Temperature, Velocity, and CFD etc.

Introduction

Heat Treatment may be characterized as warming and cooling operations connected to metals and compounds in strong state in order to acquire the coveted properties. Heat treatment is at times done incidentally because of assembling procedures that either heat or cool the metal, for example, welding or framing. Heat treatment is regularly connected with expanding the quality of material, yet it can likewise be utilized to refine the grain size, alleviate inward stretch, to enhance machinability and formability and to restore flexibility after a chilly working process. A percentage of the goals of warmth treatment (heat treatment) are abridged as takes after.

In most warmth treating (heat treatment) procedures, when materials are warmed, they respond with air gasses. In the event that response is undesirable, the work must be warmed in the vicinity of a few gas alternately gas blend other than ordinary air. This is done in ordinary environment heater handling. The gas or gas blend may be changed to bring about attractive responses with the material being handled or it might be balanced so that no responses happen. At distinctive temperatures, diverse responses may happen with the work and heater air. In most environment heaters it is impractical to change the air piece quickly enough for ideal responses or to control the climate synthesis with the degree of accuracy required for some warmth treating procedures. Vacuum heaters permit gas changes to be made quickly in light of the fact that they contain gasses of low weight. Vacuum heater innovation evacuates a large portion of the segments connected with ordinary environmental air before and amid the warming of the work.

Xuegang Lia et al. [1] they design to estimate flue gas side execution of a refining vacuum furnace with floor gas burners. Computational fluid dynamics (CFD) approach was used to replicate the flow, heat transfer, combustion and no emission. Here standard k-e model is used for turbulence simulation. Laminar flame let model is used to detect the discrete transfer model flames and NO emission. Discrete transfer model was put into practical use to the radiative heat transfer simulation. Comparative simulation approach is applied to determine the effect of more air amount on the flue gas temperature distribution and no emission. Using commercial package ANSYS CFX 14.0 calculations are done. A.Mochida et al. [2] paper they develop the method to resolve disappearing with time characteristics of the radiative and conductive heat transfer in the industrial furnaces, in vacuum furnace heat transfer is numerically calculated. Vacuum furnace is heated with many radiant tube burners and it is closed with thermal insulation walls. Object materials which is to be heated are kept at the centre
of the furnace. To solve the problems of radiative heat transfer in the enclosure along with transient conductive heat transfer in the insulation walls covered within the systems us general purpose three-dimensional computer program, and for the radiative heat exchange calculation Monte Carlo method is used. Then experimental results are compared with numerical simulation results. The validity of the present simulation method is shows when the experimental results fit well with the simulated results. O. Macchiona et al. [3] study some of the applications need high speed uniform velocity flows and with minimum pressure drop they are Gas cooled quenching and others. Flow ducting geometry is very complex if it contains following components they are circular-to-rectangular cross section, 90-180 bends and flow splitting. In forced convection furnaces same situations is present. To give design guidance in ducts and focusing on circular to rectangular ducts, to validate the results, flow was design and computed. Now result is successfully verified. With the primary geometric parameters values, Reynolds number and pressure quality of being sensitive of the velocity and pressure drop is verified in the standard range and final objective is used generate optimal design. If we increase the AR above 1.5 it may cause increasing non uniformity and pressure drop. To keep AL>1 and AR<1.5 circular to rectangular duct transitions is used. Jing Wang et al. [4] they established the computational fluid dynamics software package fluent flow and model called heat transfer model they used to simulate the high pressure gas quenching for a large H13 die. In accordance with the practical chamber set-up of vacuum furnace a method is built that is complicate geometry mesh of finite volume method (FVM). From the simulation temperature distribution of gas and velocity, and the temperature field in H13 die are obtained. Simulated cooling curves in the die compared with measured ones to validate the simulated results. Temperature dependent such as thermal properties of gas and H13 die like specific heat and viscosity it has a high effect on the accuracy of simulation results. Further it clarifies the improvement of numerical simulation.

The present study investigates in detail, the effect of mass flow rate of coolant on hot chamber inside the vacuum furnace. Under this study, 2D and 3D CAD models with the row of 6 holes at equidistance from the center line considered. These models are tested to find the flow visualization and thermal analysis of coolant inside hot chamber using CFD analysis.

Computational Model Generation (2D and 3D Models)

Model description

2D diagram shown fig.1 of vacuum furnace is designed in auto cad software. With scale 1:1 is loaded in auto cad drawing and each components dimension is measured accurately before started generating model in catia software.

![Figure 1: Sectional View of vacuum furnace chamber (2D)](image)

The model improvement is completed on CATIA shown in Fig.2. It is the preparing process. The different parts of the vacuum furnace are outlined utilizing CATIA. The parts are exclusively made and all are amassed. The profiles are produced with the assistance of directions accessible which have been created. After the development of the vacuum chamber with number of nozzles and afterward external body is made. Every one of these segments are at last last amassed and cross section is finished.

![Figure 2: Multi sectional View of vacuum furnace chamber (3D)](image)
The entire component is designed using CATIA software multi views are displayed to understand geometry. Each component is created and assembled. Surface is created by sheet metal tool.

2D Grid Generation

Ansys ICEM CFD is used to create a computational grid for both 2D and 3D shown in Fig. 2, 3, and 4. The hexa elements are used for whole domain. Mesh quality is obtained above 0.4. In total 37,957 nodes and 37,349 elements are obtained for the entire geometry. The boundary conditions are defined as given. Nozzle inlet is taken as Velocity inlet and furnace outlet is taken as the pressure outlet. Inlet is flowed with gas and mental tray also placed inside the chamber. Entire structure is meshed and the nozzles and the surface are taken as solid.

3D Grid Generation

The tetrahedral elements of T-grid mesh is constructed for hole domain. The preprocessor, solver and post processor modules are employed by ansys 13 Fluent. Mesh quality is obtained above 0.4. In total 2,07,497 nodes and 10,61,595 elements are obtained for the entire geometry. The boundary conditions are defined as given. Nozzle inlet is taken as Velocity inlet and furnace outlet is taken as the pressure outlet. Inlet is flowed with gas and mental tray also placed inside the chamber. Entire structure is meshed and the nozzles and the surface are taken as solid.

Table 1: Boundary conditions for 2D Model

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>300 k</td>
</tr>
<tr>
<td>Velocity</td>
<td>0.1 m/s</td>
</tr>
<tr>
<td>Pressure</td>
<td>0 pas</td>
</tr>
<tr>
<td>Wall thickness</td>
<td>0.01 m</td>
</tr>
<tr>
<td>Working piece boundary wall temperature</td>
<td>1200 k</td>
</tr>
</tbody>
</table>
Table 2: Boundary Conditions for 2D Duct in Hot Chamber

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Trail 1</th>
<th>Trail 2</th>
<th>Trail 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>300 k</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Velocity</td>
<td>11.87 m/s</td>
<td>0.1 m/s</td>
<td>10.5 m/s</td>
</tr>
<tr>
<td>Pressure</td>
<td>0 pas</td>
<td>0 Pas</td>
<td>0 Pas</td>
</tr>
<tr>
<td>Duct inside</td>
<td>600 k</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mass flow rate</td>
<td>1.1326 kg/s</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3: Boundary conditions for 3D model hot chamber

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Trail 1</th>
<th>Trail 2</th>
<th>Trail 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parameter</td>
<td>Temperature</td>
<td>Velocity</td>
<td>Pressure</td>
</tr>
<tr>
<td>Trail 1</td>
<td>300 k</td>
<td>0.1 m/s</td>
<td>0 pas</td>
</tr>
<tr>
<td>Trail 2</td>
<td></td>
<td>0.1 m/s</td>
<td>0 Pas</td>
</tr>
<tr>
<td>Trail 3</td>
<td></td>
<td>10.5 m/s</td>
<td>0 Pas</td>
</tr>
</tbody>
</table>

Results and Discussion

Results and Discussion for 2D Hot Chamber

Thermal performance results of flow of gases in hot chamber is using analytical approach are discussed. A further discussion on SST and CFD calculations for different cases of hot chamber is provided.

Figure 6: Pressure contour for 2D hot chamber and Velocity flow visualization for 2D hot chamber

Initial pressure value is set for zero Pascal and with flow velocity of 0.1 m/s as the temperature decreases pressure goes on increase as amount of coolant inside hot chamber increases. Pressure near the nozzle and also near the work bench is maximum around 0.023 pas. Initial velocity of nozzle flow is set for 0.1 m/s and the flow near nozzles is maximum and when it move towards the work chamber its loses its velocity. As for this above condition work bench is placed near left side of the hot chamber. Flow of velocity of coolant is maximum on left side rather than the right side. From above experiment it’s noted that 0.1 m/s is not sufficient for overall cooling. To increase the efficiency of cooling velocity of coolant in nozzle as to be increase.

Figure 7: Temperature contour of 2d hot chamber

Above figure shows the temperature profile of vacuum furnace hot chamber. Temperature in hot chamber is around 300k and as we move towards work job temperature will be maximum of 1200 k as shown in above contour.

Results and Discussion for 3D Hot Chamber

Figure 8: Temperature contour of 3D hot chamber with velocity flow through nozzle is 10.5 m/s

Above figure shows the temperature profile of vacuum furnace hot chamber but this time visualization on 3D hot chamber. Temperature in hot chamber is around 300k and as we move towards work job temperature will be maximum of 1200 k as
shown in above contour. Temperature goes on increase as it move away from nozzles. Inside the work bench maximum temperature can be visualized as show in this contour.

Figure 9: Pressure contour of 3D hot chamber for 0.1 m/s

Initial pressure value is set for zero Pascal and with flow velocity of 0.1 m/s as the temperature increase and pressure goes on increase as amount of coolant inside hot chamber increases. Pressure near the nozzle and also near the work bench is maximum around 05.608*10^-0.3pas. It's clear that pressure near outlet value is minimum say zero pas cause its closed and pressure near inside value is maximum as coolant is moving through nozzles.

Figure 10: Flow visualization of velocity contour of 3D hot chamber for 0.1 m/s

Initial velocity of nozzle flow is set for 0.1 m/s and the flow near nozzles is maximum and when it move towards the work chamber its loses its velocity. As for this above condition work bench is placed near left side of the hot chamber. Flow of velocity of coolant is maximum on left side rather than the right side. From above experiment it's noted that 0.1 m/s is not sufficient for overall cooling. To increase the efficiency of cooling velocity of coolant in nozzle as to be increase.

Figure 11: flow visualization of velocity contour of 3D hot chamber for flow velocity 10.5 m/s

Flow of gas through nozzles increase from velocity 0.1 to 10.5 m/s to know the overall efficient flow of coolant in hot chamber. Initial velocity of nozzle flow is set from 0.1 m/s to 10.5 m/s from above case study. As from this result we can conclude that now flow of coolant is efficient than previous case.

Flow of coolant is properly flowing across all chambers to increase the overall performance of cooling. Workbench is cooling with 10m/s of velocity. Velocity is maximum near the nozzle its around 10.64 m/s. So with 10.5 m/s of velocity of coolant flow is maximum and efficient.

Conclusion

CFD method for this analysis is mainly focused or used to study flow and interaction between quenching gas (nitrogen) and quenched parts or work bench in the hot chamber.

Velocity flow and pressure distribution in the vacuum furnace hot chamber is provided from CFD modeling is used to analyses the non-uniformity of heat transfer that occurred on heat treatment process.

With increase of flow velocity through nozzles inlet from 0.1 m/s to 10.5 m/s it's observed that flow is efficient and workbench cooling rate is increased. Temperature distribution can be minimized with increase of flow velocity though inlet nozzle. Pressure
inside the hot chamber increases as the flow of velocity increase. The flow field in the vacuum furnace by using nitrogen gas as coolant heat transfer is reduced to optimum value under normal pressure and high velocity. With CFD result the increase in velocity from 0.1 m/s to 10.5 m/s the heat transfer rate is decreases and it increase the furnace efficiency from 3 to 6 %.

Acknowledgements

The authors wish to thank Dr. Rana Pratap Reddy Ph.D Principal, GAT Bengaluru for permitting this work to be done at Lakshmi Vacuum Technologies Peenya, Bengaluru. The authors wish to thank Mr. Shekar, Plant head and Technical direct head LVT Bengaluru, and Venkatesh, Design Engineer LVT Bengaluru. Authors would like to thank all the members who have directly or indirectly helped us in carrying out this work.

References

[1] Xuegang Lia, Luhong Zhanga, Yongli Suna, Bin Jianga, Xingang Lia, and Jun Wan "provided Numerical simulation of the flue gas side of refining vacuum furnace using CFD".

[2] A. Mochida, K. Kudo, Y. Mizutani, M. Hattorp and Y. Nakamura "provided one more method that is Transient heat transfer analysis in vacuum furnaces heated by radiant tube burners".


[4] O. Macchion, S. Zahrai and J.W. Bouwman "developed another gas quenching method that is Heat transfer from typical loads within gas quenching furnace".

[5] Jing Wang, Jianfeng Gu, Xuexiong Shan, Xiaowei Hao, Nailu Chen and Weimin Zhang "provided a method that is Numerical simulation of high pressure gas quenching of H13 steel".

[6] Jaroslav Mackerle "provide a method Finite element analysis and simulation of quenching and other heat treatment processes".

[7] Xiaowei Hao, Jianfeng Gu, Nailu Chen, Weimin Zhang and Xunwei Zuo "provided a method 3-D Numerical analysis on heating process of loads within vacuum heat treatment furnace".

[8] O. Macchion, N. Liorb, and A. Rizzic "provide a method Computational study of velocity distribution and pressure drop for designing some gas quench chamber and furnace ducts".

[9] Olivier Macchion, Fax’enlaboratoriet "CFD in design of gas quenching furnace".

Author’s details

1Assistant Professor, Mechanical Engineering, Global Academy of Technology Bengaluru, Karnataka, India.
Email: gururaj.lalagi@gat.ac.in

2Research Scholar, Mechanical Engineering, Global Academy of Technology Bengaluru, Karnataka, India.
Email: a4adarsha4loving@gmail.com

3Professor, Mechanical Engineering, Global Academy of Technology Bengaluru, Karnataka, India.
Email: mathad_vedavyasa@rediffmail.com

4Professor & Head, Mechanical Engineering, Global Academy of Technology Bengaluru, Karnataka, India.
Email: msrajagopal@gat.ac.in