INVESTIGATION OF HEAT TRANSFER COEFFICIENT USING CFD SIMULATION IN QUENCHING PROCESS

Aulus Roberto Romão Bineli, aulus@feq.unicamp.br
Adriana Munhoz Viadana, adrianaviadana@hotmail.com
School of Chemical Engineering, University of Campinas – UNICAMP P.O. Box 6066, 13083-970, Campinas-SP, Brazil

André Luiz Jardini, jardini@feq.unicamp.br
School of Chemical Engineering, University of Campinas – UNICAMP P.O. Box 6066, 13083-970, Campinas-SP, Brazil

Rubens Maciel Filho, maciel@feq.unicamp.br
School of Chemical Engineering, University of Campinas – UNICAMP P.O. Box 6066, 13083-970, Campinas-SP, Brazil

Abstract. *The metals industry has been influenced by a constantly evolving world economy. Recently, concerns about the industry's impact on the environment have emerged as an important driving factor. As a result, render necessary optimize process to guaranty product quality, price and sustainability, through continuous innovation, improvements to the material properties and workability of engineering metals extend the range of applications in which they can be employed and the ease with which they can be manufactured. One the most important processing in metallurgic business is the metal quench, that consists in to raise steel temperature until austenitizing temperature and cooling continuously, fast or not to obtain the transformation of austenite into another desired structure, such as martensitic, ferritic, bainitic or pearlitic. For this reason, in water metal quenching it is necessary knowledge about heat transfer coefficient because it determine cooling rate that occur in this process consequently the type of material. Additionally, the fluid dynamic conditions (type of agitation) which the material is submitted in cooling, and the quenching severity, affects the properties of the final product. If a non-uniform cooling of the steel the material can also have different characteristics in the regions cooled at different speeds. However, because of the multifaceted aspects of hydrodynamics add further difficulty to the study experimentally and understanding heat transfer. In this work, the aim is simulate this process and evaluate heat transfer coefficient through in ANSYS CFX to understand the cooling of the material. The results shown that the cooling of material was not uniform and its necessary many studies to achieve a good heat extration for a unifomity cooling in steel block. This analysis was discussed and provides information in how this software can improve control of quench process bay studies of optimal quench uniformity.*

Keywords*: computational fluid dynamics, quenching, heat transfer coefficient, ANSYS CFX*

1. INTRODUCTION

Recently, the interests of the industries on the environmental impacts have emerged as an important driving factor in their productive activities. As a result, it is necessary to optimize the manufacturing processes to ensure product quality, solve time, price and sustainability, through continuous innovation. The improvements in the material properties and workability of engineering metals extend the range of applications in which they can be employed and the ease with which they can be manufactured (Ansys, 2009).

One the most important processing in metallurgic business is the metal quench that consists in to raise steel temperature until austenitizing temperature and cooling rapidly to avoid undesirable internal microstructure as well as to ensure uniform mechanical properties and minimize residual stresse. Hot metal parts are quenched using air, water, oil, or liquid polymers to obtain certain hardness and mechanical properties requirements. The main challenges face by the industry is to maintain a uniform quench rate together with a required agitation near all the surfaces of all the parts, because a non-uniform cooling would lead to residual stresses and then to warpage of the parts (Fluent, 2009).

The cooling rate is a function that depends of the liquid used in a process and degree of agitation. A simple flow modeling inside the quench tank including the parts quickly reveals non-uniform flow on certain parts, and hence non-uniform quenching. A uniform cooling can be obtained by changing the layout of the tank or geometry of hot metal parts used in the simulation.

The solution of real engineering problems through numerical techniques is now a reality in academic and industrial plants. A growing in exponential scale of modern computers, is enabling increasingly complex problems can be solved through numerical techniques. Another factor that also contributed to this trend is related to the project cost, which makes it possible that hours of testing laboratories at high costs are replaced by simulations on computers, reducing the costs and left the testing laboratory only for the refinements of the project (Veersteg and Malalasekera, Maliska, 1995, 2004).

In general, to solve a particular problem in engineering may be used analytical, numerical, experimental methods and a combination of them. The experimental method has the great advantage because use the real configuration. However, often the cost is high and can not be performed by security issues or the difficulty in reproducing the actual conditions. Additionaly the costs to carry out a large number of experiments may be prohibitive. Often, problems solved by analytical methods require that assumptions be made which deviate the real physical phenomenon or allow only geometry and simple boundary conditions. Nevertheless, analytical solutions are extremely useful to validate limit cases of numerical experiments. In turn, the numerical methods have not restrictions and can resolve problems with complicated boundary conditions and arbitrary geometries, presenting results with good speed. However, the data must be reliable as well as the numerical procedure to solve the equations.

Herewith, the numerical method CFD (Computational Fluid Dynamics) has been an integral part of many engineering projects in several industries, which request ability of predict performance of old and new projects before they are implemented. The application of CFD in analysis of fluid flow is diverse, for example: metallurgy, environment, aerodynamic, automotive, aerospace, biomedical, flow machinery, power generation, etc. This tool helps in understanding of how the fluids flow and which are the quantitative effects of their interactions in the problem.

Therefore the objective of this paper is analyze, with ANSYS CFX, the fluid dynamic conditions of the tank and heat transfer coefficient to understand the uniformity of metal cooling.

1.1. Numeric simulation CFD

Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation of fluids (Versteeg and Malalasekera, 1995). To obtain a numerical solution to any physical problem, initially, it is required the ability to create the corresponding mathematical model. The model should be resolved in non-prohibitive time and the results have to be able to represent adequately the considered phenomenon (Maliska, 2004). It is worthwhile mentioning, however, that the results generated by the CFD code are good as much as the physical (and chemical) embedded in it (Versteeg and Malalasekera, 1995). Thus, the user must have competence in several areas (computing, numerical methods, mathematics and physics of the problem and a reliable set of data and process information) for the full development of the work.

A computer simulation, taking into account flow analyses, requires an investigation of data and parameters involved in the process. The quality of these data in terms of adequacy and accuracy, can determine the attribute of the final results. Because of this, users of CFD software should be very familiar with the problems which they wish to simulate (Shaw, 1992).

Obtaining a numerical solution of any physical problem requires first the adoption of corresponding mathematical model, which as a rule is described by partial differential equations (PDE's). The idea of the method is numerically solve the PDE's, replacing the existing set of derivatives by algebraic expressions involving the unknown function. The numerical approximation gives the solution in finite number of points (called nodal points) defined by the so-called grid computing, as shown Figure 2. It is expected that the greater the number of nodal points leads the numerical solution to be closer the exact solution (Maliska, 2004).

1.2. The mathematics of CFD

The differential equations which are solved express a principle of conservation and are known as continuity, momentum and energy equations. To discretize the governing equations the software ANSYS CFX makes use of an element-based finite volume method, which firstly involves discretizing the spatial domain using a mesh (Figure 2). The mesh is used to construct finite volumes, which are used to conserve relevant quantities such as mass, momentum, and energy. All solution variables and fluid properties are stored at the nodes (mesh vertices). A control volume is constructed around each mesh node and these equations are integrated over each control volume (ANSYS CFX, 2006).

- The Continuity Equation

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho \mathrm{U}) = 0 \tag{1}$$

- The Momentum Equations

$$\frac{\partial (\rho \mathrm{U})}{\partial t} + \nabla \bullet (\rho \mathrm{U} \otimes \mathrm{U}) = -\nabla p + \nabla \bullet \tau + S_M \tag{2}$$

- The Total Energy Equation

$$\frac{\partial(\rho h_{tot})}{\partial t}-\frac{\partial p}{\partial t}+\nabla\bullet(\rho \mathrm{U} h_{tot})=\nabla\bullet(\lambda\nabla T)+\nabla\bullet(\mathrm{U}\bullet\tau)+\mathrm{U}\bullet S_M+S_E \tag{3}$$

The above set of equations are solved in ANSYS CFX and it can be simplified for a given process depending on the change in density (incompressibility or compressibility), velocity (stationary or non-stationary), and temperature (isothermal or non-isothermal). If the velocity changes are not significant in a given chemical process the momentum balance is usually not considered in order to reduce to computational effort. As a result the mathematical model for these types of problems reduces to a coupled set of heat and mass balances. Though the flow field is relatively simple (laminar, stationary, and single-phase), a rather complex behavior can sometimes be found due to the interaction of mass transfer, sorption, and nonlinear reactions.

1.3. Tank design: CAD modeling

The tank design was done in CAD (Computer Aided Design) software, originally sketched with its original features, and then the assumption was made for replacement of the nozzle ejectors only for regions that represent the nozzles. This hypothesis was adopted to reduce the size of the mesh, because to represent all pipe was necessary a greater number of elements in the mesh, which would require more computational time. Additionally by the purpose of this work the phenomena that occur inside the pipe were not of interest. The Figure 1 represents the geometric model used in the simulation.

This adopted geometry was transferred by Villares Metals industry and is a part of a research in Laboratory of Optimization, Design and Advanced Control (LOPCA), located at School of Chemical Engineering in the State University of Campinas (UNICAMP).

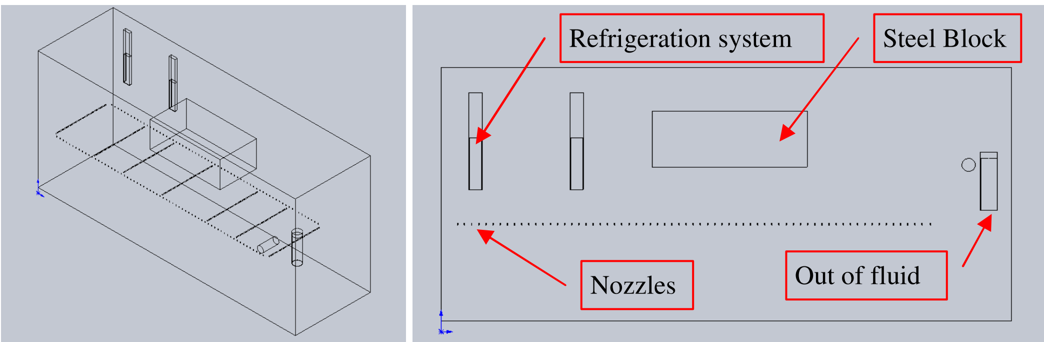

Figure 1. Geometric model used in CFX-Mesh.

1.3. Creating the geometry/mesh

The mesh generation is a fundamental step in any CFD simulation,, because the mesh must be sufficiently refined to permit desired results is achieved successfully. Therefore is advised the study the numerical mesh to obtain a sufficiently refined to be able to provide the results with the desired precision, resulting in better use of processing capacity of the computer. Care has to taken with the mesh refinement and solution computer time.

The objective of this process is to produce a mesh to serve as input to the physical pre-processor. Before a mesh can be produced, a closed geometric solid is required. The geometry and mesh can be created in CFX-Mesh or any of the other geometry/mesh creation tools. The basic steps involve:

1. Defining the geometry of the region of interest.
2. Creating regions of fluid flow, solid regions and surface boundary names.
3. Setting properties for the mesh.

In ANSYS CFX, a tank CAD geometry was imported from CAD and the mesh was generated (Figure 2). In this study the regions of ejector nozzles and block surface was refined since they are region of interest. The numerical meshes applied in this simulation shown in Figures 2 and information in Table 1.

The mesh for the model resolution was generated with approximately 10^6 elements. The quality of the mesh was verified and optimized to obtain values of Yplus (y^+) in the desired precision. Yplus is the dimensionless distance from the wall an it is used to check the location of the first node away from the wall. The software ANSYS CFX, (version 2006) recommended values of y + values are less than 100.

Table 1. Mesh information.

Number of nodes	229.295
Number of elements	
Tetrahedrons	779.294
Prisms	176.774
Total	956.068

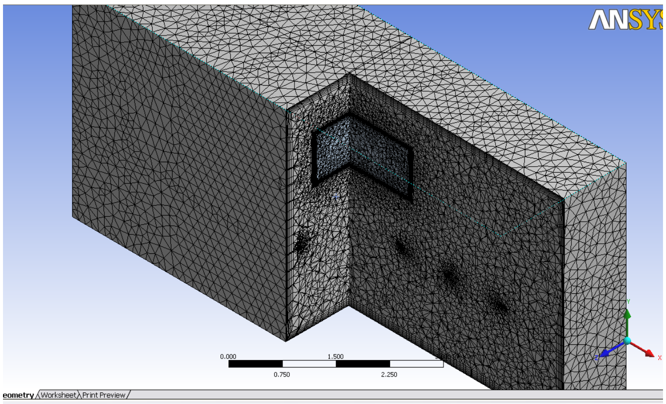

Figure 2. Isometric view with mesh cuts.

1.4. Defining the physics of the model

The mesh file is loaded into the physics pre-processor, CFX-Pre and then the thermal properties of steel block and fluid were implemented. The fluid properties that depend of temperature are found in (Handbook of Thermal Engineering, 2000). The properties of steel block were suppressed because are industrial data.

The initial temperature of steel block was 900ºC. The temperature of ejector nozzles were 35ºC and the entrance velocity of each one was 5 m/s in direction of bottom's face of steel block (axis Y). The entry velocity of refrigerant fluid (water) was 1 m/s and temperature of 25ºC in direction of bottom's tank. The two regions of output were defined with relative pressure 0 Pa, because the velocity in the output is unknown. The surfaces of the wall tank were no slip because the fluid velocity near the wall is affected by friction effects and temperature 35ºC. The top of the tank that is in contact as atmosphere, was defined as free slip surface, which the velocity near the wall is not delayed by the effects of friction, this represents an approximation of the atmospheric condition. In this simulation the flows conditions used are turbulence and the model implemented was K-ε. The model of (Kader, 1981) is used by ANSYS CFX to calculate heat flux and heat transfer coefficient at wall.

2. RESULTS

The criterion of convergence of variables associated with the momentum, heat transfer and turbulence was achieved with 10^{-4} in conjunction with a maximum of 10 iterations for each time.

The convergence associated with the turbulence model has been reached about 10^{-5}, which suggests that the implemented k-ε model was appropriate. Further, the residuals associated with heat transfer values achieved about 10^{-6}, which also suggests that the heat transfer problem has been well solved.

2.1. Cooling analysis and heat transfer coefficient

The cooling analysis and heat transfer coefficients of the block was made with the calculations of the averages of the variables of each face of the block. Figure 3 and Table 2 shown the identification of each one used. The average estimate was made by means of expressions (CEL). As noted below the values of temperature and heat transfer coefficients vary at each side.

The results of cooling profile in the Figure 4 indicate that the face 4 (below), directly exposed to the ejectors nozzles cool faster than the others, and the face 2 (upper) cool more slowly. Nevertheless, the model showed that this submerged agitation system not promotes a good uniformity of cooling curve in the quench process and the final temperature difference was 100 ° C for all sides.

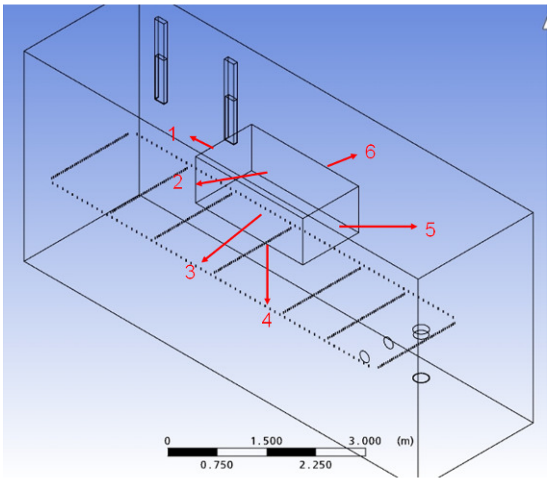

Figure 3. Faces identification.

Table 2. Faces identification.

2D Regions (faces)	
Identifications	Face's name
1	Face 1
2	Face 2
3	Face 3
4	Face 4
5	Face 5
6	Face 6

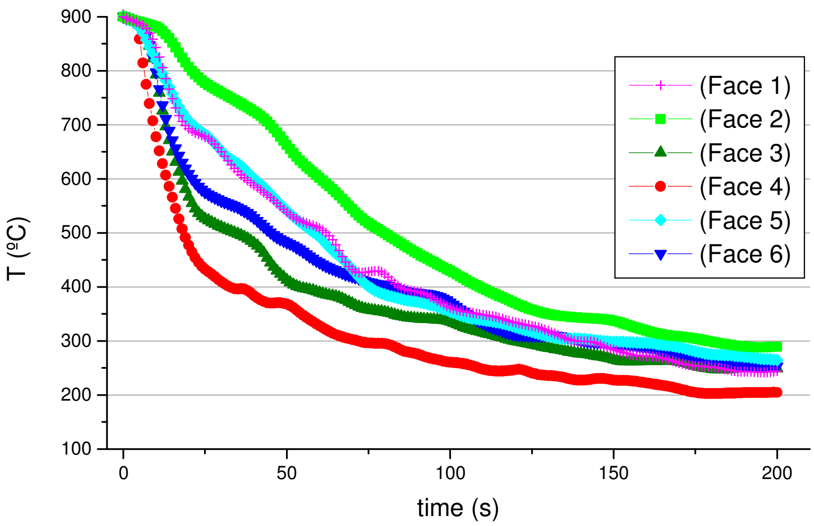

Figure 4. Cooling profile of each face of steel block.

Alongside, the heat transfer coefficients (Figure 5) showed to be consistent with the profile of cooling. The Face 4, showed higher values, around 2,6 kW/m^2 °C, and the others with lower values. This shows the relationship between the cooling and heat transfer coefficients.

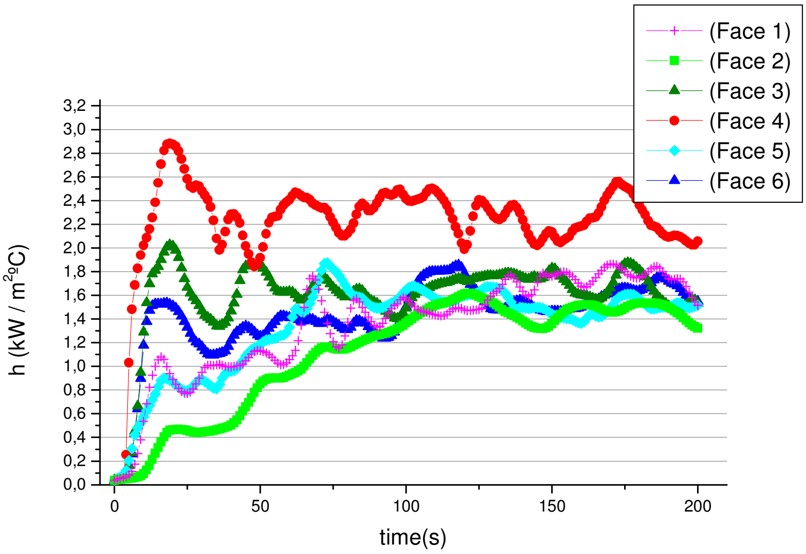

Figure 5. Heat transfer coefficient averaged of each face versus time.

The graph (Figure 6) shows the average velocity versus time calculated by CFX for each face of the block. There is a similarity in the velocity curves with the curves of heat transfer coefficients (Figure 5). This behavior is most evident in the curve of Face 4, showing a relationship between them almost direct. This demonstrates the dependence of the speed and relevance of the fluid in the cooling of the material. The higher the speed increases the heat transfer coefficients.

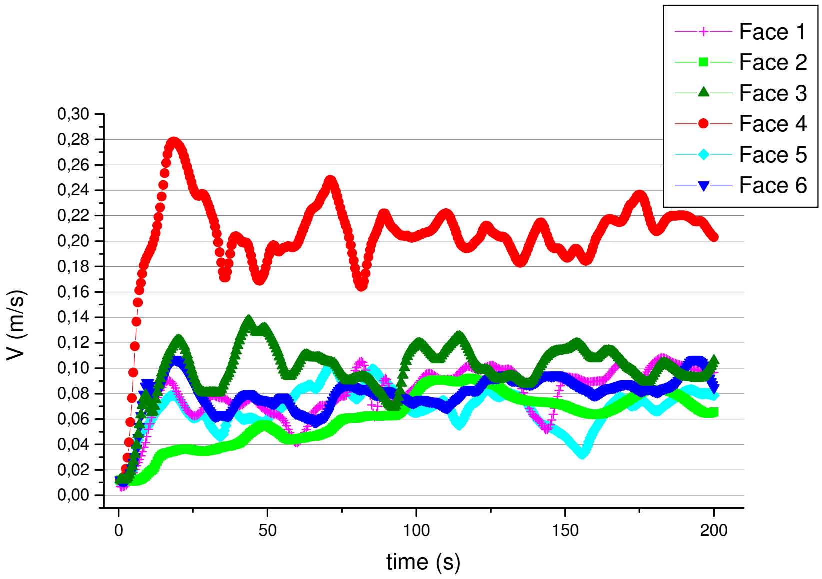

Figure 6. Fluid velocity in adjacent face of the steel block versus time.

2.2. Tank temperature

Another important factor in the cooling of any material on heat treatment is temperature of fluid refrigerant. In this analysis was possible to check the temperature of the fluid in the tank. As the temperature is also a factor that influences the value of "h", the Figure 7 show an increase in average temperature of the tank, also illustrated by Figures 8 and 9. Moreover, they also show the extraction of heat from the block and the distribution of temperature in the tank, which follows the trend to mix.

With these phenomena emphasize the importance of an efficient system for cooling the tank, besides a good heat exchanger system to control the fluid temperature for a uniform and the best as possible heat extraction in the material.

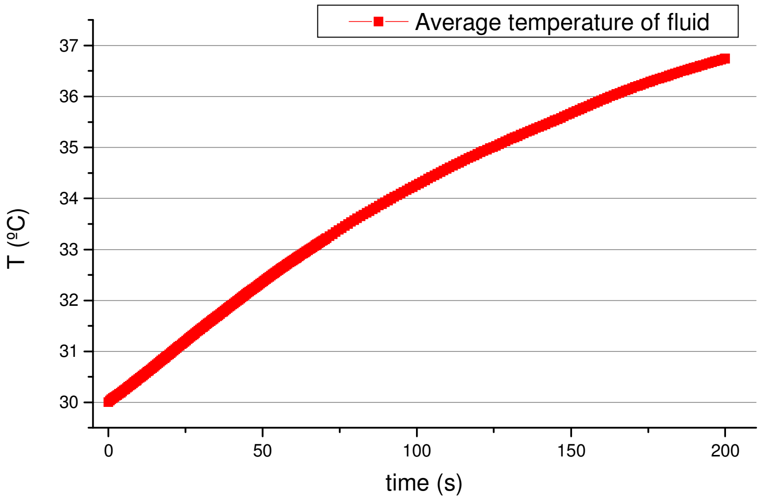

Figure 7. Average temperature in tank versus time.

The Figures 8 and 9 shows the distribution of temperature in the tank, the temperature is higher at the top and side of the steel block, which the fluid is less dense and viscous, in addition, the agitation system forces the fluid to stay on top of the tank.

The agitation system evaluated can be improved with additional agitation systems, or even exchange it for another that generates a more uniform rate. This will change the values of the coefficient of heat transfer, the cooling curves and the distribution of temperature in the tank.

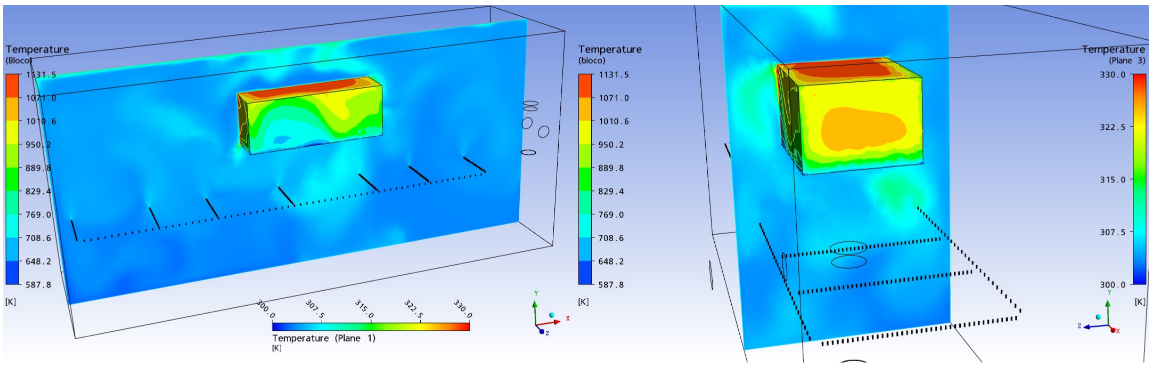

Figure 8. Temperature distribution in tank (40s).

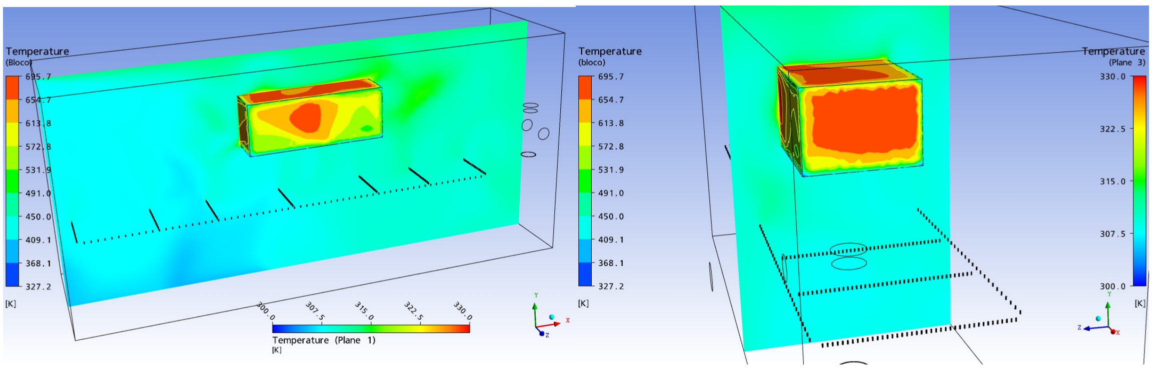

Figure 9. Temperature distribution in tank (200s).

3. CONCLUSIONS

This simulation permits to study the fluid flow behavior and steel block cooling. The use of CFD analysis to examine fluid flow in quench tanks helps the engineers improved quench system designs. The advantage of CFD analysis is that system analysis can be performed without building prototype systems.

This analysis was discussed and provides information that this software can improve control of quench process by studies of optimal quench uniformity. This approach yield significant improvements in material hardness and the use of this tool can provide an innovative design concepts for a new quench system design and the performance of the various designs.

4. REFERENCES

ANSYS. Engineering Simulation Solutions for the metals Industry. 29 April 2009.<www.ansys.com/assets/brochures/metals-industry.pdf>

Canale, L.C.F., Totten, G.E. Quenching Technology: A Selected Overview of the Current State-of-the-art. Materials Research, Vol. 8, No. 4, 461-467, 2005

FLUENT. Metal Quenching. 29 April 2009. <http://www.fluent.com/solutions/metals/metalquenching.htm>

Kader, B.A., Temperature and concentration profiles in fully turbulent boundary layers, International Journal of Heat and Mass Transfer, v.24, n.9, p.1541-1544, September 1981.

Kreith, F. Handbook of Thermal Engineering. Boca Raton, EUA: CRC Press. 2000. 1170 p.

Maliska, C. R. Transferência de Calor e Mecânica dos Fluidos Computacional. 2ª ed. LTC editora. Rio de Janeiro. 2004. 452 p.

Manual ANSYS CFX. Versão 11. ANSYS Inc., 2006.

Shaw, C.T. Using Computational Fluid Dynamics, Prentice Hall, 1992. 325 p.

Versteeg, H.K., Malalasekera, W. An introduction to Computational Fluid Dynamics (The Finite Volume Method). Prentice Hall, 1995. 257 p.

5. ACKNOWLEDGEMENTS

The authors wish to acknowledge the CNPq (National Council for Scientific and Technological Development) and Villares Metals Industry.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.