J. Energy Conversion
Volume: 12, Issue:4, 2026: 1-13
نشریه علمی-تخصصی تبدیل انرژی (JEED)
دوره ۱۲، شماره ۴، زمستان ۱۴۰۴، ص: ۱-۱۳

Numerical study on the effect of torch geometry on the combustion process inside an Copper melting anode casting furnace

Ali Afshar[1], Mohammad Hassan Nobakhti [1,*]

1-Department of Mechanical Engineering, Sciences and Researches Branch, Islamic Azad University, Tehran, Iran

Extended Abstract

Introduction

Copper anode casting is an important stage in the copper refining process, and anode furnaces are among the critical units in copper production plants. Any malfunction in these furnaces can interrupt upstream and downstream operations and consequently impose considerable production and maintenance costs. In recent operation of an anode furnace at the Sarcheshmeh Copper Complex, severe burnout and damage were observed at the burner head shortly after commissioning. Since burner-head burnout can result from excessive temperature near the burner tip and premature flame formation, modification of the burner geometry can be an effective approach to improving combustion performance and reducing thermal damage. Experimental trial-and-error design of a new burner is, however, time-consuming and costly. Therefore, the present study numerically investigates the effect of burner-head geometry on the combustion flow and temperature distribution inside an anode casting furnace. Computational fluid dynamics (CFD) simulations were employed to identify the main cause of burner-tip burnout and to develop a modified burner geometry capable of reducing the temperature in the vicinity of the burner head while maintaining a more uniform temperature distribution inside the furnace. The study was conducted using ANSYS Fluent and the finite volume method with a three-dimensional unstructured computational mesh.

2. Materials and Methods

The investigated system consists of a horizontal cylindrical anode furnace with a diameter of approximately 4.3 m and a length of 14.9 m. The gas burner is installed at the upper-left side of the furnace, and a non-premixed mixture of natural gas and air enters the combustion chamber through the burner. The existing burner has a gas-pipe diameter of approximately 14.3 cm, with 20 fuel nozzles having a diameter of 8 mm and a central opening with a diameter of 78 mm. In addition to the existing configuration, two modified burner-head geometries were developed and numerically evaluated. In Design 1, the number of fuel nozzles was increased to 48, while their diameter remained 8 mm. In Design 2, 48 fuel nozzles were also employed, but their diameter was reduced to approximately 5.12 mm. The computational domain was discretized using an unstructured three-dimensional mesh. Due to the small dimensions of the fuel nozzles and the high expected temperature and velocity gradients in the vicinity of the burner head, finer elements were applied in these regions. A mesh-independence study was conducted by comparing the static flame temperature along the furnace centerline for four different meshes containing 1,499,086, 2,260,956, 1,127,492, and 883,672 cells. Since the results obtained from the third and fourth meshes were almost identical, the fourth mesh, with 883,672 cells, was selected as the optimum computational mesh.

The simulations were performed under steady-state conditions using ANSYS Fluent R2022. The flow was assumed to be viscous, Newtonian, incompressible, and three-dimensional. The fuel and air were considered to enter the furnace as a non-premixed mixture, and complete stoichiometric combustion was assumed. The governing conservation equations for mass, momentum, energy, chemical species, turbulence, and radiation were solved using the finite volume method. A turbulence model based on the (k-\varepsilon) approach was employed to describe the turbulent flow, while the Eddy Dissipation Concept (EDC) was used to model combustion. The fuel and air mass flow rates were set to 0.0547 and 0.9416 kg/s, respectively. The furnace wall temperature was specified as 300 K, and the outlet was defined as a pressure outlet with a pressure of 10325 Pa.

3. Results and Discussion

The numerical results demonstrated that premature flame formation in the immediate vicinity of the burner head is the principal cause of the observed burner-tip burnout. In the existing burner configuration, the flame develops very close to the burner head, producing a high-temperature region that exposes the carbon-steel burner to severe thermal loading. Carbon steel experiences significant degradation of its properties at temperatures above approximately 1100 °C, while oxidation becomes considerably more severe at high temperatures and burner-tip burnout can occur at temperatures above approximately 1420 °C. The numerical results for the existing geometry indicated a maximum temperature of approximately 1700 K (about 1426 °C) near the burner head, which explains the experimentally observed damage after a relatively short operating period. Modification of the burner-head geometry significantly altered the location and development of the flame. Increasing the number of fuel nozzles and modifying their dimensions delayed fuel–air mixing and consequently postponed flame formation away from the burner head. For Design 1, the maximum temperature in the region surrounding the burner head decreased to approximately 1200 K, corresponding to a reduction of about 500 K compared with the existing configuration. For Design 2, the temperature in this critical region decreased by approximately 900 K relative to the existing burner. These results indicate that modifying the fuel-injection arrangement can substantially reduce the thermal load imposed on the burner head.

The comparison of the temperature fields inside the furnace also showed a clear difference between the proposed designs. Although Design 1 successfully reduced the temperature near the burner head, its flame shifted upward and resulted in a less uniform temperature distribution within the furnace. In contrast, Design 2 produced a considerably more uniform temperature field while maintaining a flame position closer to that of the original burner. Moreover, the maximum flame temperature inside the furnace decreased by approximately 90 K compared with the existing burner. This indicates that Design 2 provides a better balance between reducing the local thermal load at the burner head and maintaining an appropriate temperature distribution throughout the furnace. The numerical comparison presented in the study further confirmed the effectiveness of the modified designs. Based on the reported results, the maximum temperature in the burner-head region decreased by approximately 31% for Design 1 and approximately 47% for Design 2 compared with the existing configuration. In addition, Design 2 generated a more uniform furnace temperature distribution and reduced the maximum flame temperature by approximately 90 K. These improvements demonstrate that the geometry of the fuel-injection system plays a significant role in controlling the mixing process, flame location, and thermal loading of the burner.

Practical Implementation and Validation

Considering the numerical results, Design 2 was selected as the final burner configuration. Because the specific geometry of the burner and the industrial characteristics of the furnace limited the availability of directly comparable experimental studies, validation through an identical laboratory-scale experimental investigation was not available. Nevertheless, the proposed burner geometry was manufactured in the workshop of the Sarcheshmeh Copper Complex and installed in the anode furnace. Subsequent routine monitoring of the burner during operation showed that the burner-tip burnout problem was completely eliminated. Therefore, the numerical predictions were supported by successful industrial implementation, demonstrating the practical effectiveness of CFD-based burner redesign for solving thermal-damage problems in industrial combustion systems.

4. Conclusions

In this study, the effect of burner-head geometry on combustion flow and temperature distribution inside an anode casting furnace at the Sarcheshmeh Copper Complex was numerically investigated using ANSYS Fluent. The primary objective was to determine the cause of burner-tip burnout and develop a modified burner capable of reducing the excessive temperature near the burner head. The results showed that premature flame formation close to the burner tip was the principal mechanism responsible for the observed damage. Two modified burner geometries were therefore evaluated through three-dimensional CFD simulations. The results indicated that increasing the number of fuel nozzles and modifying their diameter delayed the formation of the flame and shifted the high-temperature region away from the burner head. Design 1 reduced the temperature in the critical region by approximately 31%, whereas Design 2 achieved a reduction of approximately 47% compared with the existing burner. Design 2 also reduced the maximum flame temperature inside the furnace by approximately 90 K and provided a more uniform temperature distribution than both the existing burner and Design 1. Accordingly, Design 2 was selected as the final configuration. The modified burner was subsequently manufactured and installed in the industrial furnace, and operational monitoring confirmed the complete elimination of burner-tip burnout. The findings demonstrate that numerical CFD analysis can provide an effective and economically practical approach for optimizing industrial burner geometry, reducing thermal damage, and improving the reliability and operational continuity of copper anode furnaces.

Key words: burner tip burnout, furnace combustion temperature, burner geometry modification

*corresponding author: m.nobakhti@srbiau.ac.ir

Cite this article as: Ali Afshar, Mohammad Hassan Nobakhti, Numerical study on the effect of torch geometry on the combustion process inside an Copper melting anode casting furnace. **Journal of Energy Conversion, 2026, 12(4), 1-13**.