Multi-Jet Impingement Cooling of a Hot Flat Steel Plate

Md Lokman Hosain\textsuperscript{a,b,*}, Rebei Bel Fdhila\textsuperscript{a,b}, Anders Daneryd\textsuperscript{a}

\textsuperscript{a}ABB AB, Corporate Research, SE - 721 78, Västerås, Sweden
\textsuperscript{b}Mälardalen University, School of Business, Society & Engineering, P.O. Box 883, SE-721 23, Västerås, Sweden

Abstract

One of the most important steps to control the quality in steel hot rolling is the Runout Table (ROT) Cooling. In this investigation, the heat transfer of water jets impinging on a hot flat steel plate was studied under temperatures below the boiling point to understand the convection heat transfer phenomena which is a major step preceding the boiling. Single axisymmetric jet and a pair of interacting jets are simulated using Computational Fluid Dynamics (CFD). The RANS model under steady and transient conditions as well as the \(k-\epsilon\) turbulence model are used for both 2D axisymmetric and 3D simulations. The water flow rate influence on the jets cooling characteristics is investigated.

Two sets of boundary conditions, constant temperature and constant heat flux were applied at the surface of the steel plate and evaluated. The single jet numerical results are successfully compared to published data based on measurements and analytical models. The two jets thermal performance was found to be unaffected because the jets are too far from each other to generate any additional thermal interaction.

1. Introduction

The liquid jet impingement is an effective way for cooling used in many applications because of its capacity to transfer very high heat fluxes. The cooling problem of the steel strip at the ROT is a special case of jet impingement cooling of a flat plate. Many different techniques are used to cool steel plates at ROT. For instance, impinging water at different angles with different types of nozzles, sprays, liquid curtains etc. Among them the liquid impinging jet perpendicular to the plate with circular nozzle (often called liquid bars) are common because of their robustness and high heat transfer rates. Moreover,
Impinging air or water cooling techniques are very useful for many other applications like cooling of combustion engines, electronic microchips, etc.

Cooling the hot flat strip under the ROT conditions is very complicated and often dealt with using experiments and empirical models. This application associates the strip movement to the boiling phenomena and consequently becomes very difficult to model. In this work we decided to separate the involved phenomena and focus mainly on the jets interaction as well as the effects of the liquid flow rate, excluding by that the surface movement and the phase change.

<table>
<thead>
<tr>
<th>Nomenclature</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k$</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>turbulent dissipation rate</td>
</tr>
<tr>
<td>$u_0$</td>
<td>inlet velocity</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$Z$</td>
<td>distance downward from the nozzle</td>
</tr>
<tr>
<td>$r$</td>
<td>radial distance</td>
</tr>
<tr>
<td>$d$</td>
<td>nozzle diameter (30mm)</td>
</tr>
<tr>
<td>$T_s$</td>
<td>surface temperature</td>
</tr>
<tr>
<td>$T_{s0}$</td>
<td>free stream fluid temperature</td>
</tr>
<tr>
<td>$T_f$</td>
<td>fluid temperature at inlet (20°C)</td>
</tr>
</tbody>
</table>

Two type of jets are used for cooling: free surface liquid jet and submerged jet. In this study we will focus only on free surface liquid jet. Figure 1-a shows a typical configuration of an axisymmetric jet. The zone beneath the jet is called stagnation zone which is the most important zone where most of the complex physics associated with a very intense cooling occurs. The radial flow zone is further divided into the different regions depending on the film, the momentum and the thermal boundary layer thicknesses, see fig. 1-b. The liquid film thickness can significantly vary for the laminar and the turbulent flow. The velocity of the liquid decelerates with increasing radial distance from the impinging point and an important phenomenon called hydraulic jump is observed.

![Fig. 1. (a) Impinging jet (b) Different zone of an Axisymmetric impinging jet [1], where, Region I: The Stagnation zone, Region II: The laminar boundary layer where the momentum boundary layer $\delta$ is smaller than the liquid film thickness $h(r)$, Region III: The momentum boundary layer reaches the film surface, Region IV: This is the region from transition to turbulent where the momentum boundary layer ($\delta$) and the thermal boundary layer ($\delta_T$) both reach the liquid surface, Region V: The flow is fully turbulent. $r_{mx}$, $r_{n0}$, $r_l$ and $r_h$ are radius of region I-IV respectively.](image-url)
3. Method and Results

In this chapter we present the simulation method for both the single and the two-jet cases as well as the results of their heat transfer performance compared with the experimental and theoretical results from literature. We used the Volume of Fluid (VOF) method based on the Realizable $k$-$\varepsilon$ turbulence model. We assumed a uniform liquid velocity inlet $u_0$ with constant flat plate surface temperature or heat flux.

3.1. Single Jet

The water / air interface for inlet water velocity varying from 1 to 5 m/s is shown in Fig.2. The decrease of the jet diameter due to gravity ($g$) is obtained in accordance with the theory: $D/d = 1/\sqrt{(1 + 2g(Z - z_0)/u_0^2)}$. The hydraulic jump is observed at a radial position of 500 mm for an inlet velocity of 1 m/s ($Re_d = 3 \times 10^4$). Fig. 2-b shows a good agreement between Liu et al. [2] model and the simulated liquid film height for different Reynolds numbers. The single jet was simulated using both 2D axisymmetric and 3D approaches and full agreement of the results was obtained for both boundary condition types applied at the hot flat surface.

The Nusselt number at the stagnation point is found to be in the same range as the correlations established in literature:

- Steven and Webb in [3] : $Nu_0 = 2.67Re_d^{0.57}(z_0/d)^{-1/30}(U_f/d)^{-1/4}Pr^{0.4}$
- Joo, P. H. in [4] : $Nu_0 = 0.93Re_d^{1/2}Pr^{0.4}$
- Lui et al. in [2] : $Nu_0 = 0.787Re_d^{1/2}Pr^{1/3}$

![Fig.3. (a) Nusselt number at stagnation point (b) Nusselt number for different speeds](image-url)
3.2. Two Neighboring Jet

The simulated jets are similar and separated by a distance around 14 times their diameter. The 3D mesh is presented in Fig. 4-a where only one jet is modeled because of symmetry considerations. The simulation results are compared with the single jet results. The Nusselt number plot in Fig 4-b shows that the results are similar and confirm that the distance separating the jets is too large to induce any differences in heat transfer characteristics. This jet configuration was chosen as the initial case study because it is most met in steel plants. This confirms that the jets in the ROT are often acting like isolated and not optimized to take advantage of the flow interactions. The future steps of this work will be considering smaller distances between the jets and varying flow rates to bring some lights to jet-jet interactions and ROT architecture optimization.

4. Conclusion

This work shows that computational fluid dynamics can model liquid jet dynamics and jet-jet interaction using the volume of fluid method to track the interface. The results are in agreement with the published models that are considering the far flow because the impingement zone involves very high gradients and is very complex. It is also found that two jets separated by 14 times their diameter act as isolated in terms of fluid flow and thermal characteristics. This shows that the ROT most used setting may be far from optimum. The presence of visible cold spots during our plant visits confirms the strong non uniformity of the temperature distribution on the strip surface. In a near future work the CFD analysis will be extended to include cases with smaller distances between the jets and assess a good ground for a well posed optimization.

5. References


Biography

Md Lokman Hosain is a doctoral student at Mälardalen University, Sweden. He holds master’s degree in Scientific Computing from Royal Institute of Technology, Sweden and master’s degree in Applied Mathematics from University of Dhaka, Bangladesh. His research interest is modelling complex fluid flows, combustion processes and heat transfer using computational fluid dynamics.