Ninth International Conference on Computational Fluid Dynamics (ICCFD9), Istanbul, Turkey, July 11-15, 2016

ICCFD9-xxxx

Hydrodynamics of Twin Water Jets Impingement on a Flat Horizontal Moving Surface

S. Al-Qash, M. M. Seraj & M. S. Gadala*

Department of Mechanical Engineering, University of British Columbia, Vancouver V6T1Z4, Canada
* Also: Mechanical Engineering Department, Abu Dhabi University, Abu Dhabi, UAE

Corresponding author: gadala@mech.ubc.ca

Abstract: The interaction of twin round free-surface water jets impinging vertically on a horizontal moving surface is numerically studied. Jets flow rate (15, 22, 30 L/min) and substrate velocity (0.6, 1.0, 1.5 m/s) were systematically changed relevant to steel cooling process. Realizable k-ε turbulent model and volume-of-fluid method were used to obtain the impingement flow characteristics that occur over moving surface during the simulations such as wetting front and wall jets interaction. Different types of twin jets interaction were captured depends to plate-to-jet velocity ratio. As the jets flow rate increase and/or the surface speed decreases, the wall jets arrive earlier to the interaction zone with higher momentum and up-wash fountain created. But, a thick calm dome-shape interaction type was occurred in case of low flow rate and slow plate speed where the wall jets collide together with low momentum. Different flow velocity distributions are found respect to the jets interaction type. The numerical results were agreed with the experiments.

Keywords: Twin Jets, Computational Fluid Dynamics, Moving surface, Jets Interaction.

1 Introduction

Due to the distinguishable ability of water jets impingements in removing high heat load, it has been widely used in industrial cooling. For example, cooling by water jets is utilized in paper producing, turbine blade manufacturing and metal making processes [1-3]. Arrays of water jets are also used in rapid cooling hot metal strips (run-out table, ROT) in metal making processes. In steel production, ROT cooling plays a crucial role in managing the mechanical and metallurgical properties of the final product. However, most of steel making industries still rely on trial and error method in controlling the cooling process of the red hot strip due to the complexity of both heat transfer and fluid fields over fast moving surface. Generally, most published studies were focused on the heat transfer aspect and the associated hydrodynamics part received little attention. In addition, many researchers conducted their experiments within small scale in accordance to the laboratories' condition e.g. studying the heat transfer of single water jet impingement over a stationary surface; see for example [1, 4-5].

When a single round water jet impinges on a fixed surface, the impingement water spreads radially symmetrical around stagnation point and creates a circular wetting zone. The impingement flow may experience a sudden rising of film thickness if circular hydraulic jump (CHJ) occurred at the wetting front (WF). At impingement zone, the jet normally hits the horizontal plate surface at impingement point. The incoming jet stream becomes stagnant and deflected and then, the radially start linearly spreading parallel to the surface until the radial velocity reach to impingement velocity. Further downstream in parallel region, water flow velocity is steadily decreasing due to surface drag and finally retarded impingement flow is accumulated at the edge and WF is developed and established similar to a CHJ. However, the circularity phenomenon is not seen at upstream and downstream of impingement point over moving solid wall due to the different motion directions of water stream respect to the moving substrate. Wetting zone is restricted at upstream by moving surface where wetting front (WF) demarking wetted and dry zones but water flow is accelerated by surface motion downstream. Over moving surface, impingement flow momentum is opposed by the shear frictions due to surface drag which decelerates the flow velocity upstream of jet impingement. The majority of computational studies of liquid jets mainly investigate a single submerged jet impingement over a fixed wall while the free-surface jet type receives little attention (see for example Fujimoto et al. [6, 7] and Tong [8]). Fujimoto et al. [9] experimentally and numerically studied the flow structure of single round liquid jet over a moving surface that is covered by a film of water (plunging jets). They observed three types of flow structures, namely, steady flow, transition, and unsteady flow, depends on the experimental conditions. Nevertheless, a single jet is not enough in removing such high and uniform amount of heat from large surface and, in turn, sets of water jets arrays are necessary to cover entire plate width and improve the overall cooling rate. When extra jet is placed, different flow structure is seen due to jets interaction. Figure 2 shows the flow structures of twin water jet impinging on a stationary surface in in jet-to-jet plane. Five fundamental regions are free jet, impingement region, inner wall region, outer wall region, and interaction zone (Int-Z). Basically, regions 1-4 are same as for single jet case but region 5 is a unique feature of twin jets case which happened due to collision of the inner wall jets and creating an upwash fountain. This causes complexity and makes it a transient and unsteady flow problem (i.e. fully turbulent). Hence, the flow structures over moving surface is totally different from fixed surface and is more complex.

The studying of hydrodynamics of water jets impinging on unheated moving surface can be considered as the first step to study the overall flow characteristics in the cooling process. Despite the importance of multiple water jets impingement on a moving surface to the industrial problem, a few studies took it on their considerations particularly various fluid structures happened. Among those little studies, Kate et al. [10] experimentally studied the flow interaction of two impinging water circular jets over fixed plate and found types of interaction by varying spacing between the two jets. The two identical water jets meet at midway distance and upwash interaction was created. Fujimoto et al. [11] studied the flow structure due to twin circular water jets over a moving surface that is covered by a water film (plunging jets case). The experiments captured that the water flow was stable and very thin at the vicinity region near to the jet impact points. The flow transferred to transient mode and subsequently to unstable mode when the two HJs meet each other and form a fountain and the flow becomes fully unsteady eventually. Both heat transfer and hydrodynamics aspects of long multiple water jets impinging over heated [2, 12-14] and unheated [15-16], were studied by UBC ROT group at a pilot-scale ROT facility as an ongoing project during years. In terms of examining the fluid field relevant to ROT problem, they studied impingement of single water jet (10 - 45 L/min) on different plate speeds (0.3 - 1.5 m/s) to

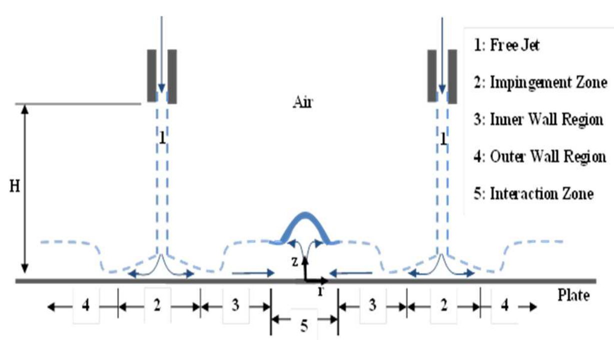


Figure 1: Schematic of twin water jets impingement over a stationary plate

replicate somehow the situation of moving long strip [17-18]. The experiments were recorded by camcorders using high speed mode. By systematic change of jet and moving surface parameters, they illustrated main characteristics of flow structures and regions over a moving long test plate: distortion of the circular wetting front into the noncircular shape, delineation of upstream surface into non-wetted and wetted regions and different flow structure at downstream. In addition, effect of jet-to-plate velocity ratio on place and width of WF (HJ) was explored and a correlation was proposed for maximum radius of wetting zone. in addition, Seraj [15] tested twin and triple jets impingement on moving long substrate through extensive experiments. The number of nozzle (N), jets flow rates (Q) and nozzle spacing (s) were varied to create variety of flow interaction in Int-Z. The type of wall jets interaction depends on the velocity ratio which includes calm thick dome-shape and very chaotic and unsteady fountain like as the unique feature for multiple jets. Numerically, Seraj et al. [16] investigated the impingement flow of single and twin circular long water free-surface jets over a stationary plate and single jet on moving surface [17]. Two different turbulent models i.e., realizable k-ε model (RKE) and the shear-stress transport k-ω model (SST) were utilized and the numerical results validated with their experimental data. Both turbulent models showed good performance but in case of SST model, near wall region required extra mesh refinement which, in turn, increases the number of cells and makes the computation so lengthy. Thus, in case of twin jets, they used RKE turbulent model in order to control the huge number of cells. As such, the objective of this study is to numerically study the fluid field aspect of impinging twin water jets of flow rates of 15, 22, 30 L/min over a moving surface with speeds of 0.6, 1.0, 1.5 m/s.

2 Numerical Simulation

The hydrodynamics of twin water jets impingement over a moving surface was modelled using the volume-of-fluid (VOF) method [19]. It is a well-known multi-phase method to trace the interface of two immiscible fluids in a mixture. VOF method was utilized in all these simulations to capture the free-surface of water jet before impingement and tracking the wetting front propagation after impingement. Basically, a volume fraction scalar parameter f is assigned to each phase inside every cell and then located them into the domain where empty cell is full of air. In each control volume or cell, the total volume of fraction for both phases is unity. Therefore, the continuity equation for volume-fraction of one phase, water, for example, can be used to track the interface and then each phase will be located accordingly:

$$\frac{\partial}{\partial t}(f_w\rho_w) + \nabla.(f_w\rho_w\mathbf{U}) = 0 \qquad (1)$$

in which the subscript w indicates water phase and U refers to the velocity vector that is shared between the two phases so then one momentum equation needs to be solved for the entire domain. Properties such as density ρ and viscosity μ are defined based on the volume fraction of each phase.
In case of incompressible turbulent flow, both velocity and pressure variables are not constant and have fluctuation part (e.g. $U_i = \overline{U_\iota} + \acute{u_\iota}$ where $\overline{U_\iota}$ is the mean part and $\acute{u_\iota}$ is the fluctuating part). Therefore, the Reynolds-Average Navier-Stokes (RANS) equations with constant properties will be [20-21]:

$$\frac{\partial \bar{U}_i}{\partial x_i} = 0 \qquad (3)$$

$$\rho\frac{\partial \bar{U}_i}{\partial t} + \rho\bar{U}_j\frac{\partial \bar{U}_i}{\partial x_j} = -\frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i}\right) - \rho\overline{\acute{u_\iota}\acute{u_J}}\right] + \rho g_i \qquad (4)$$

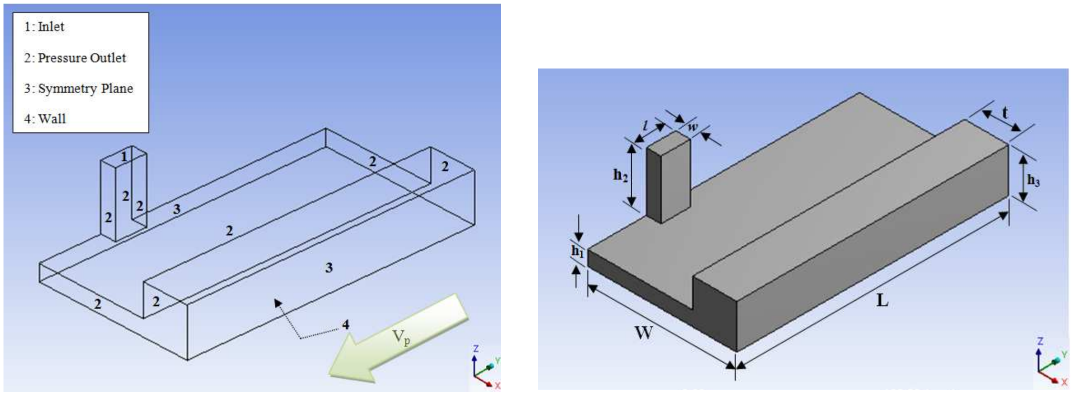
where p refers to pressure, g indicates the gravity vector, ρ and μ are density and viscosity, respectively. The term terms with (¯) indicate their mean magnitude part and terms with (´) refer to their fluctuation part. The 3-D twin water jets were solved numerically through transient RANS equations using finite volume method besides the VOF method by CFD package ANSYS FLUENT 14.5. The Realizable k-ε (RKE) model was utilized among all of the available turbulent models since it sufficiently predict the spreading rate of planer and round jets [22]. In addition, the non-equilibrium wall function [23] was used to enhance the modelling of water layer near the moving surface and saving the computational time. Actually the wall function treatment is adequate for this high-Reynolds industrial flow problem instead of studying the boundary layer. All governing PDE's were replaced by a set of discrete algebraic equation and were integrated over the entire domain. The Pressure-Implicit with Splitting of Operators (PISO) scheme was used for velocity and pressure equations in these transient simulations. The time was discretized implicitly using first order scheme. But, a second order upwind method was selected for both momentum and turbulence equations. The VOF equation was discretized using a geo-reconstruction scheme. Each algebraic equation was iteratively solved until converged solution was obtained. Generally, the normalized residual for continuity equation was set as order of (10-4) and velocity and turbulent parameters were set as order of (10-6). The constant properties for both air and water were considered at atmospheric condition: for the primary phase, air $\rho = 1.225$ kg/m^{3}and $\mu = 1.7894\text{e-}05$ kg/m.s and for the secondary phase, water $\rho = 998.2$ kg/m^3 and $\mu = 1.003\text{e-}03$ kg/m.s and finally the surface tension between the two phases is $\sigma = 0.0728$ N/m. For more details on setting up the solution procedures refer to the program user's guide [24]. Overall, all the simulations were run at a small time step in order of (10^{-5}) to ensure solution stability and until unchanged results were established.

2.1 DOMAIN, BOUNDARY CONDITIONS AND MESH

There are mainly four boundary conditions: inlet, pressure outlet, symmetry plane, and wall (Figure 2a). The half of jet and Int-Z planes were assumed as symmetry planes in order to control the huge number of total cells. Otherwise, powerful computational facility is required. The target surface is a moving wall with different speed in the negative Y-direction and no-slip condition (Figure 2b). At the inlet, the fully developed profiles of axial velocities and turbulent profiles were implemented from the previous 2-D simulations of each flow rate at 50 mm height above the plate [17]. The pressure outlet B.C. was assigned to the domain's boundaries where the flow can exit the domain but no water backflow is allowed.

The UBC ROT experiments conditions [15] were used to size the 3D computational domains for twin water jets simulations. To find appropriate dimension of the domain, the upper region (in +Y direction) was carefully designed based on parameters (jet flow rate and plate speed) of each case (Table 1). The main purpose was to minimize the domain as much as possible in order to reduce the total number of cells generated during meshing process. In all cases, the downstream region was fixed at 50 mm in -Y direction. Table 1 summarizes the dimensions for the computation domain for all the cases of flow rates over three different plate speeds. The total height of the jet inlet is considered as 50 mm (h1+ h2).

The twin jets domain was meshed with non-uniform structural grids of rectangular cells which is clustered towards the plate surface and also especially at the impingement and the interaction zones. In each case, the original number of cells was set with consideration of simulation time and required CPU memory, and then the mesh was refined progressively according to the wall y+ requirement ($y^+ = \sqrt{(\rho\tau\_w\ )}\ z/\mu$ where τw is the wall shear stress). Basically, in all of the simulations, the wall y+ was maintained in the range of $5 < y+ < 10$. Also, each simulation was monitored and checked out continuously to ensure the proper spreading of impingement flow over moving substrate especially for the lower flow rates 15 and 22 L/min jets over faster moving plate. The mesh adaptation had to avoid the water layer intermittency and so the mesh was mostly performed in the wall bounded region that included impingement water layer and the adjacent air flow. Therefore, this dramatically increases the number of cells and made the simulations so lengthy and memory consuming. The final meshed computational domain included at least 800,000 to millions of cells. The computations were conducted on powerful workstation and a 32-core workstation in parallel processing mode. Mesh dependency was studied after ensuring that the impingement water has a continuous development over the moving substrate. More details about mesh refinement and study of mesh independency are



(a) (b)

Figure 2: Three dimensional computational domain, (a) boundary conditions (b) domain

Table 1 Model Dimensions for Three Flow Rates and Three Plate Speeds

Jet Diameter, dj (mm)	Plate Speed V_p (m/s)	Flow Rate, *Q* (L/min)	Inlet (mm)	Length L (mm)	Height h_3 (mm)	Width W (mm)	Int-Z thickness t (mm)
19	0.6 1.0 1.5	15	20×10	185	30	101.6	30
		22	25×12.5	220	40		
		30	30×15	260*	40		

* The total length (upstream + downstream zones) for only 1.0 and 1.5 m/s cases

presented in [24].Then, appropriate meshes were used in each case to optimize the computation time and the numerical outcomes are considered grid independent.
In total, nine distinct cases were considered of 15, 22, and 30 L/min jets flow rates impinging on the moving surfaces with speeds of 0.6, 1.0, and 1.5 m/s. The simulations were usually continued until steady impingement flow and jets interaction settled down over moving surface. Thus, all the results are presented here are of unchanged conditions.

3 Results and Discussions

First, wetting front propagation over the moving surface is considered at the upstream region where it encounters the restrictions exerted by the plate movement. From the top view one can monitor the wetting front propagation as well as its shape. Figure 4 represents the water flow propagation of 30 L/min on a moving wall of 1.0 m/s, for example. At the very beginning, the impingement water spreads similar to L/min on a moving wall of 1.0 m/s, for example. At the very beginning, the impingement water spreads similar to impingement on a fixed plate (circular shape) in impingement zone and nearby. However, the wetting front is gradually deformed and changed to non-circular shape as propagate radially in parallel zone (after 0.025 sec). As water layer covered larger area over the plate top surface, its velocity decreased due to the frictional drag from the moving surface and the water front thickened. Also, the impingement water as inner wall jets reached to the interaction zone and produced created thick interaction water layer and then a bulge of water a head of Int-Z was formed. Figure 5 illustrates the experiment of this case (30 L/min on 1.0 m/s). Overall, the numerical result of VOF contours of water flow development and start of wall jets interaction is in good timing with the experiments. During the experiments splashing and splattering of water droplets were observed which were not reproduced much in these simulations. This can be mainly attributed to the

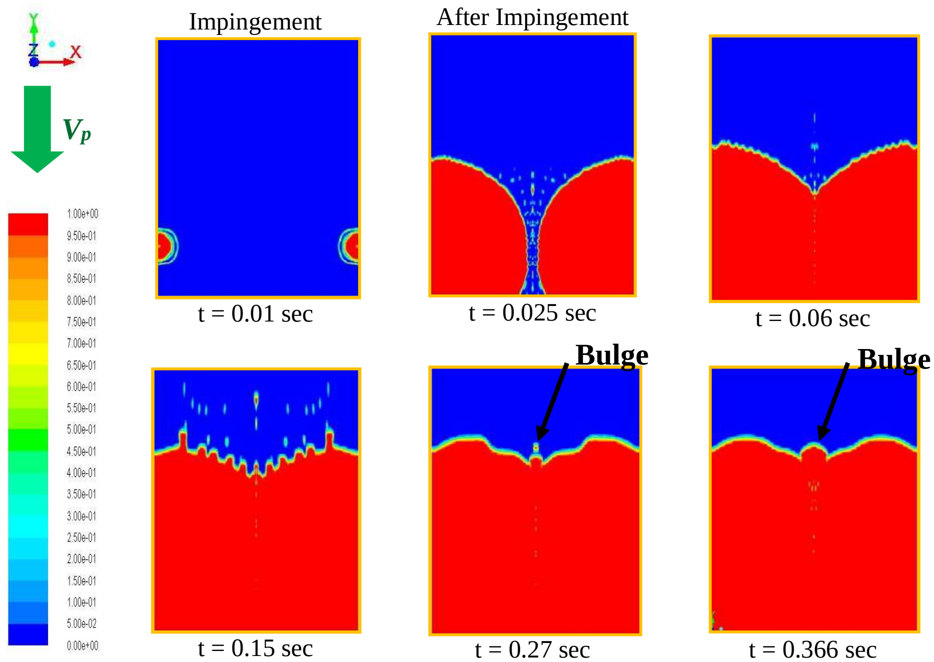


Figure 4: Twin water jets development of 30 l/min over 1.0 m/s moving surface

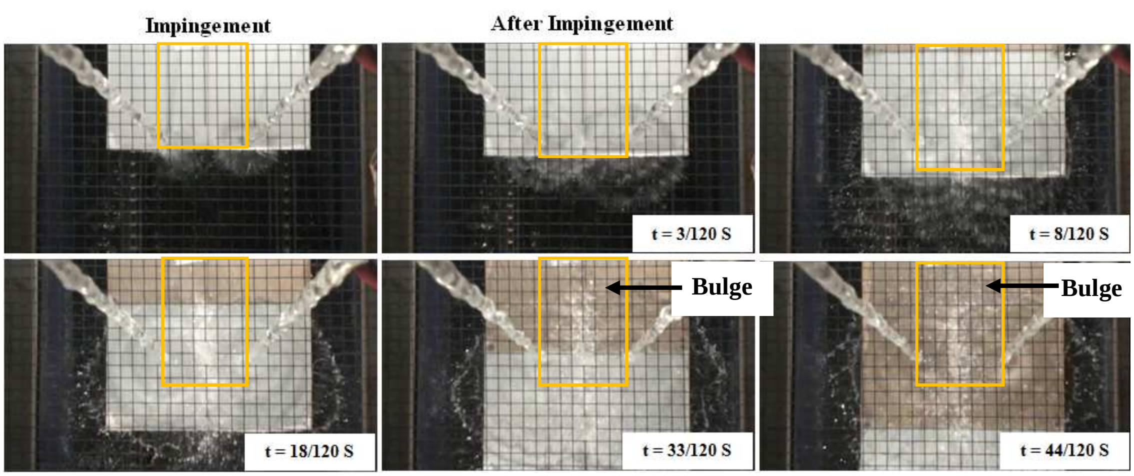


Figure 5: Sample of top view images for water flow propagation of 30 l/min over 1.0 m/s

turbulent model that being used as an averaged-based RANS model and also different surface condition of test surface and wall surface in the computational domain.

At the jet symmetry plane, not only the wetting front propagation but also the hydraulic jump (HJ) location and configuration can be detected. As shown in Figure 6, when the jets flow rate increase (Q = 15, 22, and 30 L/min) at same velocity moving plate such as 0.6 m/s, for example, the wetting front propagates further distant from the impingement point and the HJ is getting unstable and takes place accordingly with different structures. When the amount of impingement water was set to 15 L/min, the HJ location is at about 4.5dj but when the water amount is doubled to 30 L/min, the HJ settles at the double distance about 9dj. This is because the impingement waters are flowing on the moving plate with same speed and surface condition but the jets momentum is doubled. In addition, Figure 6 illustrates thin or very thin computed water layer in parallel zone outside the impingement zone. In case of 15 L/min, the water has the thinnest layer and the free-surface stands almost at z = 0.2-0.5 mm

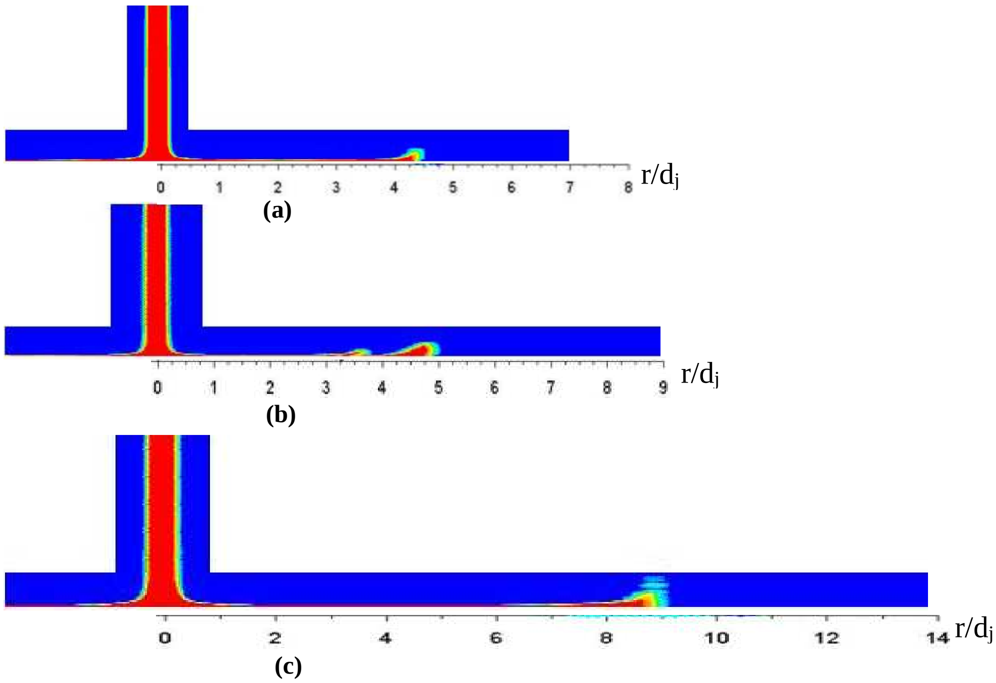


Figure 6: Wetting front over moving plate V_p = 0.6 m/s, (a) Q = 15 L/min (b) Q = 22 L/min (c) Q = 30 L/min

In case of 22 L/min, the water film has a moderate thickness where the free-surface is as high as about 0.5-1.0 mm. The 30 L/min twin jets developed not a very thin impingement water layer and the free-surface becomes higher, z = 1.0-3.0 mm. Opposite to upstream, the thin water layer became thinner radially in downstream. In fact, moving substrate put shearing effect under flow layer in flow direction. Therefore, the accelerated flow should experience the reduction in thickness. Basically, in case high jet flow rate Q = 30 L/min, the impingement water covers larger area of the wall surface in comparison to the lower flow rates 15 and 22 L/min. When the impingement water has higher momentum, it spreads out faster over the moving surface and better counteract the confinement effect of the frictional drag force on the wetting front propagation and then spreads more toward the upstream zone (in +Y direction).

At Int-Z symmetry plane, the Y-component velocities (Vy), in surface motion direction, at different elevation were obtained in order to examine the effect of moving plate on the two wall jets interaction. The velocity profiles along with jets interactions are shown in Figure 7. At downstream zone in all cases, the velocity grows in direction of wall motion (negative values) and continues to increase at upstream zone until 3.5dj or beyond where velocity magnitude became zero. Although a sharp increase in velocity magnitude is seen but the velocity is very low positive velocity, Vy/Vimp ≈ 0.04-0.05 (Figure 7a, b). Actually, the inner wall jets have very low flow velocity when enter the Int-Z while the effect of plate motion is same everywhere in − Y direction. Sharp velocity reduction specifies occurrence of wetting front and, in turn, change of water phase to air phase consistent with water contours in Int-Z. Smooth curves for velocity at different depth in Int-Z are evidence of calm interaction of 15 L/min wall jets represented by the dome-shape interaction type (Figure 7a). In cases of 22 and 30 L/min, the velocity was fluctuating at the downstream zone as a sign for chaotic flow structure due to collision of higher momentum wall jets. For 30 L/min jets (Figure 7c), a higher water layer shows the up-wash fountain kind of interaction that producing different velocity variation. Higher velocity picks (Vy/Vimp ≈ 0.1) and high rise accumulated water layer are resulted from stronger flow stream of wall jets due to 30 L/min jets while the plate velocity is same as before, 0.6 m/s Again, sharp velocity reduction marks the place of wetting front. The location of the wetting front at the Int-Z symmetry plane is less distant from inter-jet line in both cases of 15 and 22 L/min respect to the WF location at the jet symmetry plane (see Figure 6). This is due to the superiority of motion of plate respect to the flow stream at Int-Z that alters the circular shape and curves it to non-circular form. In case of 30 L/min, nevertheless, the WF has extended further upstream distance at Int-Z symmetry plane compared to jet symmetry plane, different scenario compare to 15 and 22 L/min jets. Actually after wall jets interaction takes place, a bulge of water a head of Int-Z is created which pushes the WF even further away in upstream zone (r/dj $\approx$ 11). This bulge also was observed in the experimentation.

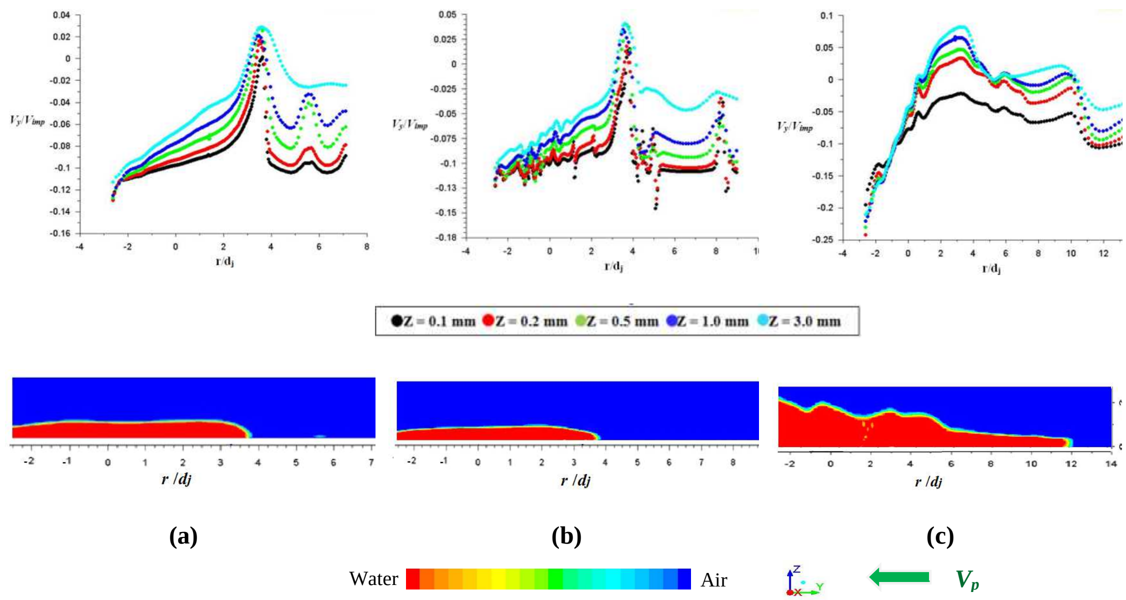
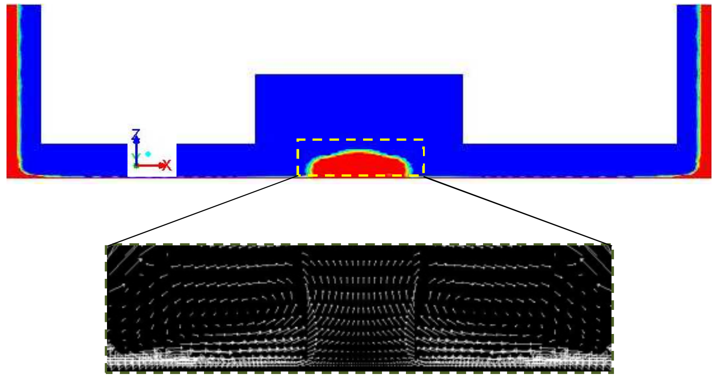
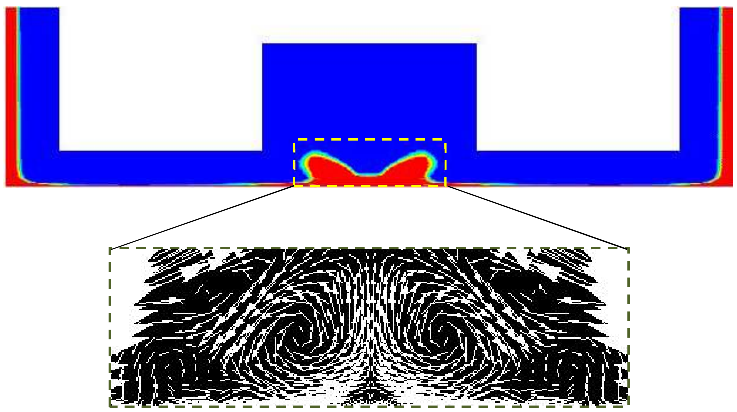


Figure 7: Y-direction velocity and jet interactions at int-z symmetry plane on moving plate V_p = 0.6m/s, (a) Q = 15 L/min (b) Q = 22 L/min (c) Q = 30 L/min

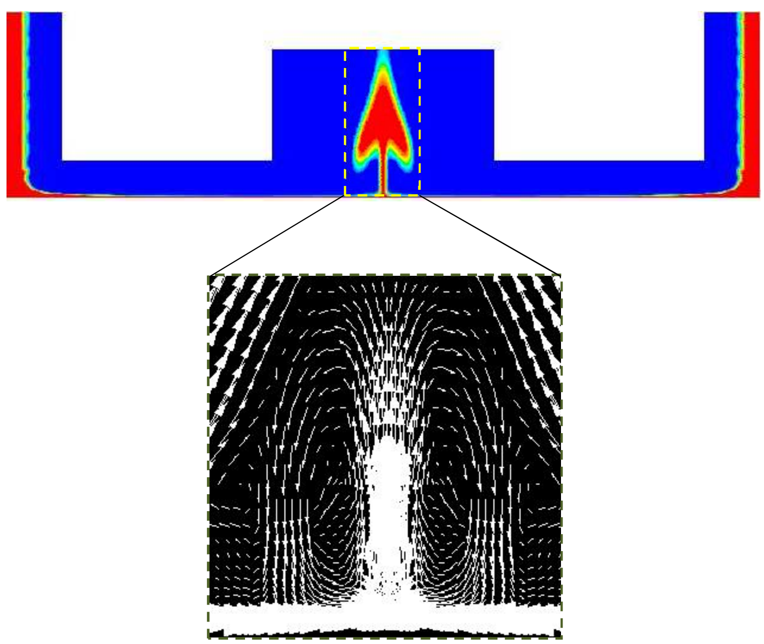
Depends on the flow rate of parent jets and plate surface speed in the simulations, different interaction types produced after the water wall jets collided in the Int-Z. Figure 8 presents various interactions of twin water jets through set of simulations with fixed plate speed such as 0.6 m/s, for example, As shown in inter-jet plane, a thick dome-shape interaction type was captured in case of low flow rate 15 L/min jets, a thin tilted fountain like interaction in case of 22 L/min jets and a thin up-wash fountain interaction type in case of the high flow rate of 30 L/min. This is in a good agreement with the UBC ROT experimental observations [15]. The flow velocity vectors were also plotted at the jets interaction and nearby region (highlighted by a dotted-yellow rectangular) to evaluate the behavior of resulted interaction. These velocity vectors clearly illustrate the behavior of flow such as circulation. When the flow rate was low as 15 L/min (Figure 8a), the resulted interaction was thick dome-shaped and velocity vector are small with smooth change in the direction signifying calm interaction type. This interaction type introduces weak circulation in adjacent air flow. In fact, the water stream from very thin impingement layer has reduced momentum when reach from impingement zone to the Int-Z due to the dominating effect of drag force of the moving wall. In cases of 22 and 30 L/min (Figure 8b, c), however, the velocity vectors are larger and have sharp change in the direction showing the chaotic flow and strong circulatory behavior which transferred to the surrounding air as well. Actually in these cases, more water and faster moving stream within the wall jets increases the water flow



(a) *Q* = 15 L/min



(b) ***Q* = 22 L/min**



(C) ***Q* = 30 L/min**

Figure 8: Different interaction types of twin water wall jets and velocity vectors on a moving plate V_p= 0.6m/s

momentum which could survive the frictional effect of moving substrate all the way down to the Int-Z. Resulted strong wall jets could create the thin and energetic and so high-rise interaction type with very unstable structure and big induced circulation as depicted in Figure 8 for 30 L/min case.

3 Conclusion and Future Work

The hydrodynamic of twin circular free-surface water jets impinging over moving surface was numerically investigated. The RKE turbulent model with the aid of non-equilibrium wall function treatment was used along with VOF method to trace wetting front propagation, hydraulic jump location and contour, and the wall jets interaction. The mesh refinement was crucial to resolve the water flow intermittency above the moving surface and to maintain continuous spread of wall jets. Faster moving surface confined more the wall jets and wetting front and hydraulic jump happened closer to the inter-jet-line all over the surface. According to different plate-to-jet velocity ratios, three distinct interaction types were captured namely, thick dome-shape, thin tilted fountain-like, and thin up-wash fountain presenting different velocity variations within interaction flow. Generally, the numerical outcomes agree well with the UBC ROT experimental observations.

References

[1] R. Viskanta, and F. P. Incropera. *Quenching with Liquid Jet Impingement*, In: Tanasawa I, Lior N (eds) *Heat and Mass Transfer in Materials Processing*, Hemisphere Publishing New York, 455-476, 1992

[2] G.A. Franco, M.A. Wells, and M. Militzer. Effect of nozzle configuration on cooling of a hot moving steel plate. Canadian Metallurgical Quarterly, 48:197-206, 2009

[3] X. Liu, L.A. Gabour, and J.H. Lienhard V. Stagnation-point heat transfer during impingement of laminar liquid jets: analysis including surface tension. J. Heat Transfer, 115:99-105, 1993

[4] D. H. Wolf, F. P. Incropera, and R. Viskanta. Jet impingement boiling. Advances in Heat Transfer, 23:1-132, 1993

[5] F. C. Kohring. Water all water-cooling systems, Iron and Steel Engineer, 30-36,1985

[6] H. Fujimoto, H. Takuda, N. Hatta, and R. Viskanta. Numerical simulation of transient cooling of a hot solid by an impinging free surface jet. Numerical Heat Transfer Part A, 36: 767-780, 1999

[7] H. Fujimoto, and S. Dejima. Numerical analysis and heat transfer of a free surface circular jet impinging onto a substrate. ASME Int. Mech. Eng. Cong. Exp., Seattle, USA, 8:1117-1124, 2007

[8] A. T. Tong. A numerical study on the hydrodynamics and heat transfer of a circular liquid jet impinging onto a substrate. Numerical Heat Transfer Part A, 44: 1-19, 2003

[9] H. Fujimoto, Y. Suzuki, T. Hama and H. Takuda. Flow characteristics of circular liquid jet impinging on a moving surface covered with a water film. ISIJ Int., 51:1497-1505, 2011

[10] R. Kate, P. Das and S. Chakraborty. An experimental investigation on the interaction of hydraulic jumps formed by two normal impinging circular liquid jets, J. Fluid Mech., 590, 2007, pp. 355-380

[11] H. Fujimoto, Y. Lee, R. Kato, T. Hama and H. Takuda. Experimental and numerical study of twin circular water jets impinging on a moving thin film, ISIJ Int., 53: 1427-1435, 2013

[12] P. Chan, Jet Impingement boiling heat transfer at low coiling temperatures, MA. Sc.

Thesis, University of British Columbia, Canada, 2007
[13] K. Jondhale, M.A. Wells, M. Militzer, and V. Prodanovic, Heat transfer during multiple jet impingement on the top surface of hot rolled steel strip, Steel Research Int., 79: 938-946, 2008
[14] G. Franco, Boiling heat transfer during cooling of a hot moving steel plate by multiple top jets, M.A.Sc. Thesis, University of British Columbia, Canada, 2008
[15] M. M. Seraj, Hydrodynamics of circular free-surface long water jets in industrial metal cooling, PhD Thesis, University of British Columbia, Canada, 2011
[16] M. M. Seraj, E. Mahdi, and M. S. Gadala, Numerical assessments of impingement flow over flat surface due to single and twin circular long water jets, Transaction on Control and Mechanical Systems, 1: 290-299, 2013
[17] M. M. Seraj, and M. S. Gadala, Noncircular hydraulic jump on a moving surface due to an impinging circular free surface jet of water, Steel Research International, 83:653-670, 2012
[18] M. M. Seraj, and M. S. Gadala, Experimental investigation of wetted zone of an impinging circular water jet on a moving surface, ASME Int. Mech. Eng. Cong. Exposition, IMECE-39399, Canada, 2010
[19] C.W. Hirt, and B.D. Nichols, Volume of fluid (vof) method for the dynamics of free boundaries, J. of Computational Physics, 39: 201-225, 1981
[20] White F. M., Fluid Mechanics, Ch. 5, USA: McGraw-Hill, 1998
[21] M.A.R. Sharif, and A. Banerjee, Numerical analysis of heat transfer due to confined slot-jet impingement on a moving plate, Applied Thermal Engineering, 29: 532-540, 2009
[22] T.H. Shih, W.W. Liou, A. Shabbir, Z. Yang,and J. Zhu, A new k-ε eddy-viscosity model for high Reynolds number turbulent flows - model development and validation, Computers Fluids, 24: 227-238, 1995
[23] S.-E. Kim, and D. Choudhury, A near-wall treatment using wall functions sensitized to pressure gradient, ASME FED Separated and Complex Flows, ASME 217: 273-279, 1995
[24] ANSYS FLUENT, User's Guide, Chapter 12: Modeling Turbulence, 2013
[25] S. I. Alqash, Numerical simulations of hydrodynamics of twin water jets impinging over a horizontal moving plate, MA. Sc. Thesis, University of British Columbia, Canada, 2015