Scientific Journal of Impact Factor (SJIF): 5.71
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406
International Journal of Advance Engineering and Research Development
Volume 6, Issue 07, July -2019

Heat Transfer Characteristics of Rotating Cylinder with Water Impinging Jet

Mitali Basargi[1] , Dr.J.G Gujar [2], Dr. S. M. Pise [3]

[1, 2] *Department of Chemical Engineering, Sinhgad College of Engineering, Pune, India*
[3] *Department of Mechanical Engineering, KIT's College of Engineering, Kolhapur, India*

Abstract —*Heat transfer in spray cooling has been studied extensively in various industrial operations such as metal quenching, hot rolling and atomization etc. The landing of liquid on metal efficiently is important part of study. Spray technology cools the hot metal by improving ability of liquid to be absorbed. Nozzles are used to distribute liquid droplets onto heated surfaces. Spray cooling occurs when liquid forced through a small orifice into dispersion of fine droplets which impact on heated surface. The droplets spread on the surface and absorb the heat there by cooling the surface.*
Most machinery reaches power conversion in the form of rotation, such as mechanical processing machine, textile machine, gas turbine, and power electronic equipment, with a wide application range. The operating temperature of rotating machinery is constantly increased due to the generation of friction heat, combustion or power output. Thus, effective cooling technology should be used for heat dissipation, in order to protect normal operation of the machine parts. The heat transfer in spray cooling is studied experimentally and validated with the CFD analysis of spray cooling.

Keywords- *Heat transfer, CFD, Spray cooling, Heated metal surfaces, low heat flux*

I. INTRODUCTION

Heat transfer in spray cooling has been studied extensively in various industrial operations such as metal quenching, hot rolling and atomization etc [1]. For optimizing mechanical properties of materials, cooling rate is important in manufacturing industries also nozzles are used to distribute liquid droplets onto heated surfaces [1]. There is necessity of in-depth knowledge of heat transfer of complex flows which includes knowledge of the interaction between the jet and the moving surface. Also the knowledge of interaction between jets and ramps is essential [2]. There are considerable studies going on with quenching of hot surface at high initial temperature up to 800-900℃ considering pipe laminar, slit laminar and flat spray jets. To understand fundamental boiling process from film to transition boiling, the characteristics of transient transition boiling heat transfer were reported [3].
One of the effective cooling technologies is known as impinging [4]. The local heat of the parts and significantly improving the heat dissipation performance. The quenching is carried out using hot moving strip by many sub cooled water jets to ensure the cooling [5]. Quenching is needed for ensuring very high cooling rate. In nuclear plants, quenching of the reactor core is a safety procedure in case of a loss of primary refrigerant accident. So it is necessary to predict accurately the rate of cooling. In manufacturing industries, quenching can be used for cooling molds or controlling the structure of the steel alloys. Structure and thus mechanical properties of steel alloys are conditioned by the cooling rate of the product.
It is thus of primary importance to well control this cooling rate and its homogeneity to obtain steels with good and homogeneous mechanical properties [6].
The knitting cylinder is a rotating cylinder during operation. While the rotating cylinder exposes the entire circumference to the impingement flow, the local heat and mass-transfer coefficients at different locations of the cylinder are more symmetrical [7]. The visualization study of air cooling is carried out for round hole nozzle in the center of the cylinder which is the part of annular channel [8]. Experiments with significant cooling rates are performed by M. Gradeck et al and the further study is needed to perform for the collapse of heat flux [9]. The new methodology is developed to carry out 3-D flow simulation using anaglyph which is used in hydrodynamic research field [10].

II. SPRAY COOLING : WORKING AND PRINCIPAL

Impinging cooling technology is one of the most common and effective cooling technologies. Its design is simple, and the cooling fluid can directly and sufficiently impinging on machine parts, thus carrying away the local heat of the parts and significantly improving the heat dissipation performance. In practical industrial applications, such as food processing, textile and paper drying process, there are heat transfer and mass transfer problems on the side of the rotating cylinder under impinging jet flow. In particular, knitting cylinder of the cylinder knitting machine for textile production may be overheated in prolonged running operation, thereby affecting the machine operation. The knitting cylinder is a rotating cylinder during operation. Spray cooling occurs when liquid forced through a small orifice shatters into a dispersion of fine droplets which then impact a heated surface. The droplets spread on the surface and evaporate or form a thin liquid film, removing large amounts of energy at low temperatures due to the latent heat of evaporation in addition to substantial single-phase convection effects. Heat transfer rates much higher than can be attained in pool boiling are possible with sprays since there is less resistance to the removal of vapor from the heated surface. Other advantages include the possibility of uniformly cooling large surfaces, low droplet impact velocity, and no temperature overshoot. Some disadvantages include the need for pumps, filters, and the need to transport excess liquid and vapor to a condenser.

III. EXPERIMENTAL WORK

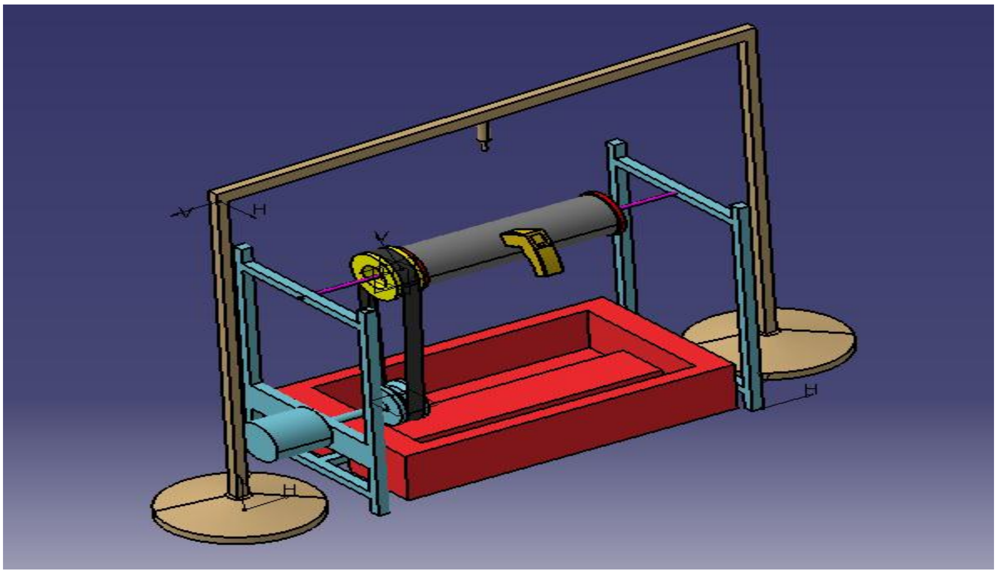

Figure: 1 Schematic experimental set up

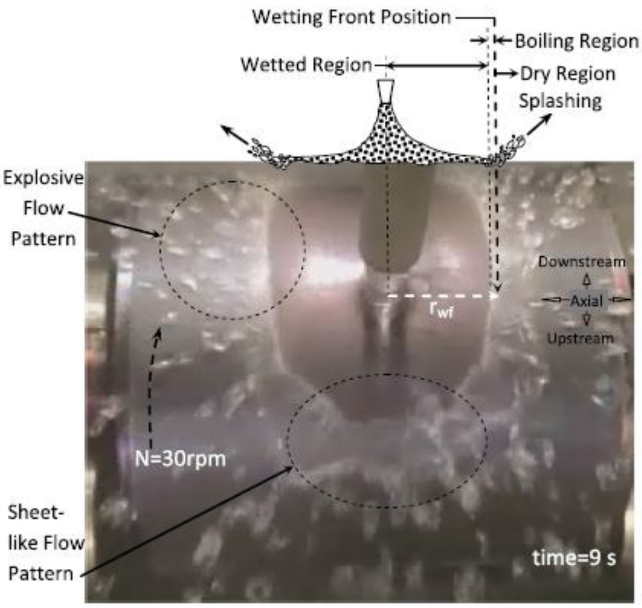

Figure 2. Spray parameter study [1].

The schematic experimental set up is shown in Figure 1.The experiments were conducted for the low heat flux application. Figure 2 elaborates the nature of the flow patterns which are seen during the experimentation. Sheet like flow pattern is observed when spray collapses away from the center point.

Figure 3: lab set up for the spray

Readings have taken for the various low temperature values to observe the effect of low heat flux on the spray cooling of the metal. The GIMP software is used to measure the diameter of the droplets while spraying. The initial temperature and final temperature after cooling measured with the temperature measurement device. Figure 3 shows lab set up for the spray analysis. The Buckingham's Pi theorem is used to obtain the correlation for the Nusselt number. This theorem is used for determining the number of independent dimensionless groups that can be obtained from a set of variables. The Table No 1 and 2 shows the Correlation values compared with experimental and correlation values. For various pressure values, the number of readings has been taken. The temperature is measured at the surface of the spray before cooling and after cooling. Using all the data, calculations are performed and correlation is developed between the dimensionless parameters. The correlation obtained is given below:

$$NU = 0.067 \times Re^{0.83} \times Pr^{0.33} \times \frac{U}{V}$$

Table 1 Experimental vs Correlation values

Nu/pr^0.33(Experiment)	Nu/Pr^0.33(Correlation)
0.082	0.080
0.079	0.081
0.078	0.079

Table 1 predicts values of Nusselt number obtained using experiments and correlation.Using correlation , heat transfer coefficient is calculated.

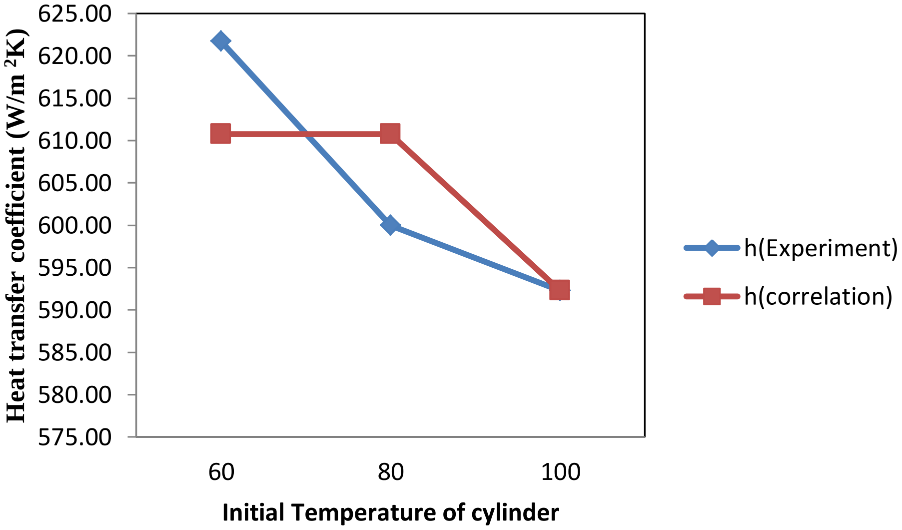

Figure 4: Graph of h experimental vs Correlation

Figure 4 shows the graph of heat transfer coefficient vs temperature values. The calculated heat transfer coefficients using Experiment, Correlation and CFD approach are given in table 2.

Table 2 Experimental data and results

Sr. No	Spray Height (m)	Pressure (bar)	Flow rate (kg/s)	Temperature (°C)	h (w/m^{2}k) (Correlation)	h (w/m^{2}k) (Experiment)	H (w/m^{2}k) (CFD)	% Error
1	0.07	2	0.005856	60	610.75	621.70	652.44	-4.95
2	0.07	2	0.005856	80	610.75	599.98	651.76	-8.63
3	0.07	2	0.005856	100	592.28	592.27	650.66	-9.86

IV. CFD ANALYSIS APRAOCH

The CFD analysis is done to validate the results obtained from the experimental data. CFD (computational fluid Dynamics) method is used to study the flow dynamics inside the system. Numerical analysis is performed using CFD approach. [11]. A stepwise process is carried out for an analysis. Firstly, geometry of spray domain which is considered as CFD domain is prepared using CAD software. Then CFD domain is discretized using meshing. To acquire approximate solution numerically, discretization method is used. In this method partial differential equation is converted in to simple algebraic equation. Finite volume method, Finite element method and finite difference method are used for discretization. The Eulerian-Lagrangian approach is used to solve the flow equations [12]. The finite volume solution method can either use a "segregated" or a "coupled" solution procedure. The segregated solution method is the default method in most commercial finite volume codes. It is best suited for incompressible flows or compressible flows at low Mach number. Droplet break up is modeled in the simulation with the discrete phase modeling. Uniform particle distribution is selected for the droplet break up modeling.

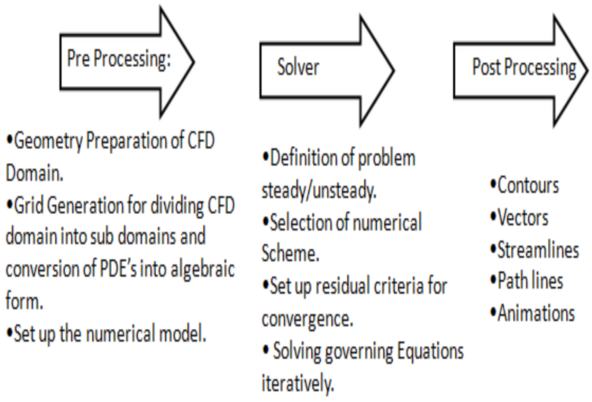

Figure 5: Steps of CFD analysis

The boundary conditions are set then simulation is solved untill convergence criteria met. Figure 5 shows the steps followed to perform CFD analysis. Also these steps are helpful for setting the boundary conditions.The boundary conditions are defined with Nozzle as an inlet and bottom surface of the tank as an outlet. The K-E turbulent model is used to solve the turbulent flow in the system of spray. To capture an effect of rotations on cylinder, multiple reference frame approach is used. The minimum and maximum droplet size is defined using experimental data. Droplet break up model is enabled. Transient simulation carried out to study an impact of cooling on rotating cylinder. Solver solves an equations of Navier stokes. In the Navier stokes equation Mass conservation, Energy conservation, Momentum balance etc equations are solved according to problem type [13].

Mass Equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{V} = 0$$

Energy Equation:

$$\rho\left(\frac{\partial E}{\partial t} + u\frac{\partial E}{\partial x} + v\frac{\partial E}{\partial y} + w\frac{\partial E}{\partial z}\right) = \nabla \cdot (k\nabla T) - \nabla \cdot p\vec{V} + \dot{Q}_v + \dot{Q}_g$$

Momentum balance equation:

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right) = -\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left[\mu\left(2\frac{\partial u}{\partial x}-\frac{2}{3}\nabla\cdot\vec{V}\right)\right]+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right]+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+B_x$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right) = -\frac{\partial p}{\partial y}+\frac{\partial}{\partial y}\left[\mu\left(2\frac{\partial v}{\partial y}-\frac{2}{3}\nabla\cdot\vec{V}\right)\right]+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right]+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right]+B_y$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right) = -\frac{\partial p}{\partial z}+\frac{\partial}{\partial z}\left[\mu\left(2\frac{\partial w}{\partial z}-\frac{2}{3}\nabla\cdot\vec{V}\right)\right]+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)\right]+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right]+B_z$$

Turbulence K.E equation is mentioned below.

$$\rho u_j \frac{\partial k}{\partial x_j} = \rho\tau_{ij}\frac{\partial u_i}{\partial x_j} + \frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] - \left[\rho\varepsilon + 2\mu\left(\frac{\partial k^{1/2}}{\partial x_j}\right)^2\right]$$

Eddy Dissipation Rate is given by

$$\rho u_j \frac{\partial \varepsilon}{\partial x_j} = C_1\frac{\varepsilon}{k}\tau_{ij}\frac{\partial u_i}{\partial x_j} + \frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_T}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] - C_2\frac{\rho\varepsilon^2}{k} + \frac{2\mu\mu_T}{\rho}\left(\frac{\partial^2 u}{\partial x_2^2}\right)^2$$

Momentum balance equation is solved for x, y and z directions. This process is solved iteratively to get solution.

V. CFD VALIDATION STUDY

The CFD study for 1mm nozzle diameter is carried out. The experimental data is used for the validation. The CFD simulation is performed to validate the temperature and velocity obtained from an experimental data.The results show the application of CFD method instead of experimental study for design of experiments. Images are taken for before cooling and after cooling. Spray behavior is observed. Initially, temperature is set to 60 °C and rotational speed is set to 120 RPM. The speed is kept constant. The initial velocity and temperature is taken in contour using CFD analysis software. Here Fluent is used to validate the results from experiment.

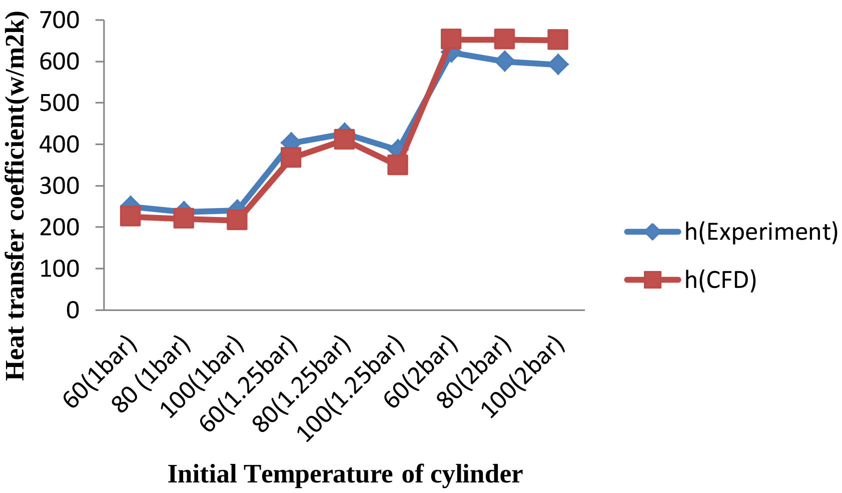

Figure: 6 Graph of heat transfer coefficient obtained from experiment and CFD

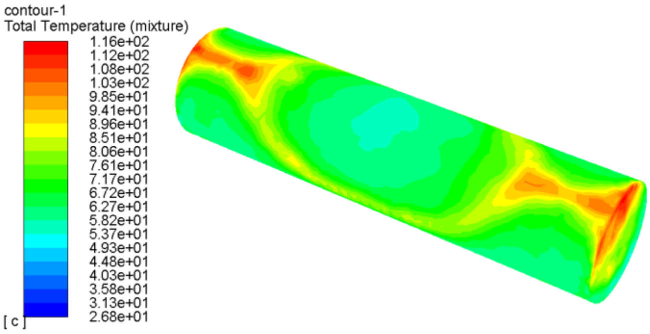

Figure: 7 Temperature contour at initial stage (isometric view of the cylinder)

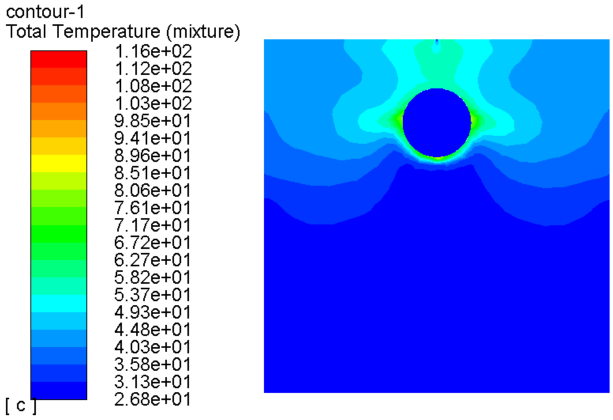

Figure: 8 Temperature profile at initial stage (Side view)

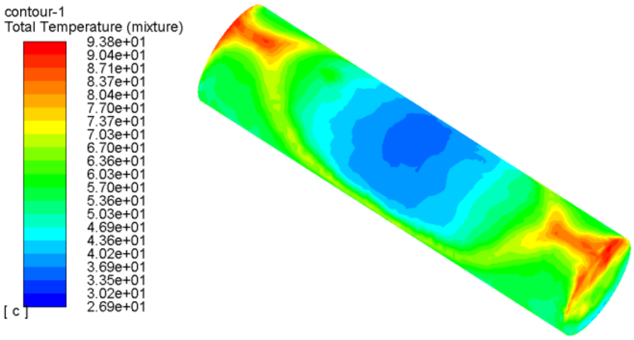

Figure: 9 Temperature profile after cooling (Isometric View)

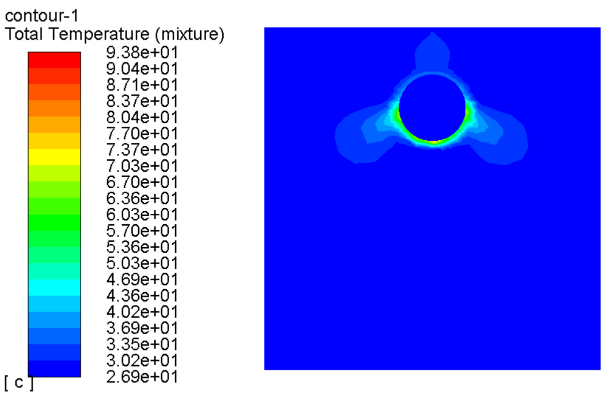

Figure: 10 Temperature profile after cooling (Side View)

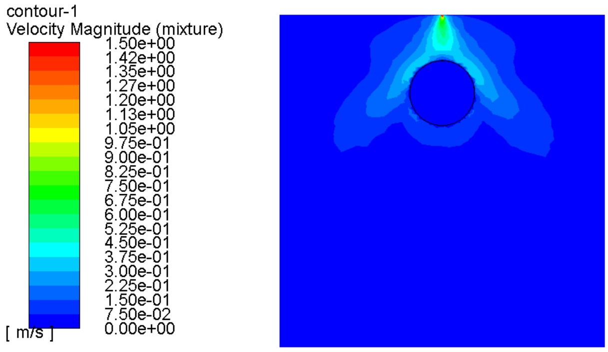

Figure: 11 Velocity contour after cooling (side view)

The temperature profile is shown in figure no.7, 8, 9 and 10 for the different view of geometry. Figure 11 shows the nature of spray spreading on cylinder. The temperature values obtained from the experiments and the CFD results are validated with $\pm 10\%$ error. Hence the validation of the experimental study is carried out using the CFD technique.

VI. CONCLUSION

Using experimental results, the correlation is developed. After developing the correlation using the dimensionless numbers, data is validated with the use of CFD analysis approach. Temperature values at the initial stage are given as an input to the CFD analysis. The CFD analysis has been performed for validation case and optimization is carried out using CFD approach. Temperature is the most critical parameter observed for the cooling. When low heat flux is studied, temperature plays more critical role while achieving the heat transfer. Cooling will be achieved at low heat flux values. Temperature gradient will be less in the low heat flux cooling. Low heat flux is critical parameter to be study. This depends upon major parameters like spray temperature, flow rate, spray height, cooling time. Spray height plays important role for achieving more desired heat transfer. Droplet dynamics is essential part to be studied. Droplet dynamics should be studied in detail while, the droplet hitting the heat transfer surface. It is critical parameter to study. Based on droplet dynamics the accuracy of the simulations will be more hence the experimental costs should be minimized. CFD analysis plays more important role in minimizing the experimental costs. Spray cooling can be achieved using brine solution instead of water as coolant. The brine solution can help to achieve the required rate of the cooling. Effective cooling technology is required to protect machinery from friction heat which is generated by rotating machineries due to industrial operations like combustion.

VII. REFERENCES

[1] Aloke Kumar Mozumder, Yuichi Mitsutake and Masanori Monde “Subcooled Water Jet Quenching Phenomena For A High Temperature Rotating Cylinder.” International journal of heat and mass transfer vol. 68, pp.466-478, 2014.
[2] M. Gradeck,A. Kouachi J.L. Borean ,P. Gardin and M. Lebouche “Heat Transfer From A Hot Moving Cylinder Impinged By A Planar Subcooled Water Jet.”International journal of heat and mass transfer vol. 54, pp. 5527-5539, 2011.
[3] Shinya Tsuboyama, Tomofumi Higashi, Yuichi Mitsutake and Suhaimi bin Illias “Characteristics Of Transient Heat Transfer And Wetting Phenomena During Laminar Jet Quenching On Rotating Cylinder”, Procedia Engineering vol. 105, pp.787 – 797, 2015.
[4] Tzer Ming Jeng, Sheng Chung Tzeng and Rui Xub “ Heat Transfer Characteristics Of A Rotating Cylinder With A Lateral Air Impinging Jet” International journal of heat and mass transfer vol.70, pp.235-249 , 2014
[5] Chen Wen long Han Feng Liu Qui Nie and Fan Han Lin “Spray Characteristics and spray cooling heat transfer in the non-boiling regime”, Energy, issue, vol.38, pp. 3399-3405, 2011.
[6] M. Gradeck a, A. Kouachi b, M. Lebouché a, F. Volle a, D. Maillet a and J.L. Borean b “Boiling curves in relation to quenching of a high temperature moving surface with liquid jet impingement”, International journal of heat and mass transfer, vol.52, pp.1094-1104, 2009
[7] Hongting Ma, Dan Du, Jinglong Sun, Yufeng Zhang and Na Deng “Convective mass transfer from a horizontal rotating cylinder in a slot air jet flow,” International journal of heat and mass transfer, vol.54, pp.186-196, 2011.
[8] Tzer-Ming Jeng, Sheng-Chung, Tzeng Jeng and Hong-Liang Chang. “Flow visualization in an annulus between co-axis rotating cylinders with a circular jet stationary outer cylinder.” International journal of heat and mass transfer , vol.39, pp.1119-1124, 2012.
[9] M. Gradeck , A. Kouachi , J.L. Borean , P. Gardin and M. Lebouché “Heat transfer from a hot moving cylinder impinged by a planar subcooled water jet,” International journal of heat and mass transfer, vol. 52 pp.1094-1104, 2009.

[10] KANG Su-hai et al “Application of video anaglyph maker for 3-D flow simulation." Journal of hydrodynamics, vol. 22(2), pp.289-294, 2010.

[11] R Panner Selvam, R Ponnappam “Numerical Modeling of nucleation boiling in the thin film and effect of droplet impact”.Annual Thermal & Fluids Analysis Workshop. TFAWS, 2004.

[12] Versteeg Malalasekara An introduction to CFD finite volume method 2nd edition. 2007

[13] Ansys Fluent 12.0 theory guide.

