International Journal of Engineering Sciences &Research Technology

(A Peer Reviewed Online Journal)
Impact Factor: 5.164

IJESRT

Chief Editor
Dr. J.B. Helonde

Executive Editor
Mr. Somil Mayur Shah

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

CFD ANALYSIS OF CIRCULAR HYDRAULICJUMP ON A CONVEX TARGET SURFACE

Nitesh Kumar*[1] **& Prof. Ashish Muchrikar**[2]
[1]M.Tech Scholar, 2Professor & Head Department of Mechanical Engineering
[1]Corporate Institute of Science and Technology, Bhopal, India
[2]Corporate Institute of Science and Technology, Bhopal, India

ABSTRACT

The main objective of the present work is to examine the consequence of changing height of nozzle inlet from a convex target surface using a 2-D axisymmetric computational fluid dynamic analysis. An initial study on straight plate is done to validate the analysis result with existing data. In the study the convex modelling of the target surface is done by parameterizing a non- dimensional number and target surface radius of 70mm and curve radius of 1000mm, 1300mm and 1600mm. The height of impingement is varied as 10, 15 and 20 mm. It is found in the analysis that, at 1000 mm curve radius percentage increase in jump radius is 19.43%. For 1300 mm curve radius percentage increase is 54.42%. For 1600 mm curve radius the value is 29.16%. Also it is found that at constant height of 10 mm of nozzle with respect to increase in curve radius from 1000mm to 1600 mm the percentage decrease of 33.32%.After analyzing all the data of simulation it is concluded that increase in the height of nozzle from target surface results in the increase of radius of circular jump

KEYWORDS: circular hydraulicjump, nozzle height, curve radius, target plate radius, jump height, dimensionless number.

1. INTRODUCTION

Every time when a fluid jet is impacted on a hard surface it blowouts centrifugally outward with definite flow characteristics. It is seen that in centrifugally outward direction after certain length a steep jump in flow occurred. This sudden jump in fluid continuum is called circular jump. This is because of transition of stream flowing through supercritical to sub critical. The implication of such flow is found in cooling area of various devices as per need. The various application area of phenomena is gas turbine blade cooling, paper industry, forging and milling operation etc. A schematic diagram for the concept is as below:

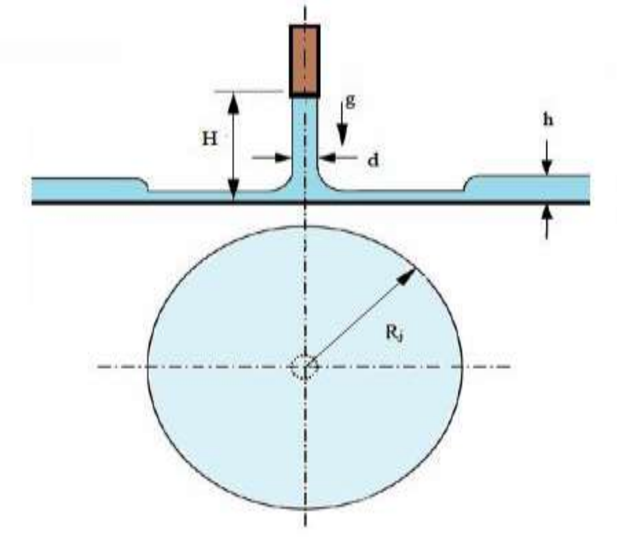


Fig. 1 Schematic diagram of circular jump

Physics of impinging jet and formation of jump

When a jet of water strikes a flat surface a quick rise in pressure take place which enforce the liquid to accelerate after incoming to the target inlet and streaming downward stream of water strike location called as stagnation point and the thin liquid flick form in directive small height (millimetre) which spread beside the centrifugal direction and environs the entire region. The friction effect which is present on plate causing a kinematic boundary layer and difference in temperature amid the fluid and target plate or surface cause a thermal boundary layer. The velocity of fluid along the superficial goes on decreasing with upsurge in distance from stagnation point which causes the liquid flick thickness and boundary layer thicker. The jet strike is configured into 3 regions. Thever is free superficial jet in which the fluid used in jet strike is less dense like an air. Secondly, flooded jet that permits the fluid to strike into identical liquid. Third restricted flooded jet in which the stream area is demarcated with the help of wall. The free surface shape is very much critical to design as it is effected by gravitation force, surface tension and pressure and this pressure depends upon shape extent and stream speed of jet. The heat transfer through a jet strike is very complex method because it involves number of parameter such as Reynolds number, jet diameter, stream rate, spacing distance i.e. space among jet and goal surface, stream velocity, and jet inclination etc.

Watson was first person who analysed the viscous circular jump. He used the boundary layer model for the upstream of the jump and presumed the drift in downstream region to be inviscid. Assuming the pressure thrust to be equal the speed of momentum destruction, he derived the following for the jump condition:

$$\frac{1}{2}g(H_\infty^2 - H^2) = (\frac{Q}{2\tau r R_j})^2\left(\frac{1}{H} - \frac{1}{H_\infty}\right)$$

where$H$ being the upstream height, H_∞ the downstream stature (or outer deepness), g the gravitational acceleration, Q the volumetric flow speed, and R_j being the radius of the jump. The final result of the inviscid theory in non-dimensional form, attained by disregarding the term containing $(HlH_\infty)^2$ in Eq. (1) and making some substitutions, is specified by

Fig. 2 The general structure of circular jump

$$\frac{R_j H_\infty^2 g a^2}{Q^2} + \frac{a^2}{2\tau^2 R_j H_\infty} = \frac{1}{7\gamma^2}$$

where$a$ is the radius of the incoming jet before impact. Watson showed that, inviscid theory is not accurate enough for predict-ing the radius of the jump by comparing his experimental results with this theory. The similar conclusion is made from numerical consequences, as deliberated later in this tabloid.

In the current case the flat target surface is replaced by convex one with existing criteria of convex parameter.

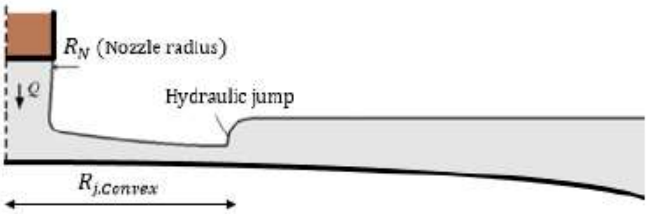


Fig. 3 Schematic diagram of a jump on convex target surface.

2. LITERATURE REVIEW

We found the number of hypothetical and experimental work of research is done on circular jump with stationary jet strike on flat surface but all the works has been conceded out with stationary jet only but moving jet are too less. Hence for this we are required to explore the hypothetical and experimental work.

Kate et al. [1]explore the jump experimentally, observed two category of this portent with upper angle of jet preference angle a level curve profile is formed and lower inclination angle profile with corner was observed. They quantize the flick thickness for diverse flow rates and with unalike jet angles. The flick thickness was measure by using conductive post-mortem for both forward and reluctant in radial direction and for before and after jump.

Ahmad Saberi et al. [2] studied the effect of curvature of the target plate first time for ethylene glycol as employed fluid and concluded that the increase in curvature increases the jump radius in case of jet strike flow on a targete plate. The height of targate plate from the position of nozzle was held at 1 cm and3unlike convex targete surface were choosen for analysis and experimental analysis was done.

Mohammad Khavari et al. [3] investigated the consequence of the impinging stream from a nozzle on the circular jump formed. A CFD learning has achieved by the authors to scale and detect the volume fraction of the water and glycol. The outcome liberated in the study that upsurge in the mass stream rate upsurges the radius of jump. For chasing the crossing point of the air and water the VOF(volume of fraction) model was used in CFD analysis. A mesh liberation study was also accomplished by the author and it was found that a value of 20 CPR(cell per unit radius) was suitable for stable radius.

Vishwanath et al. [4] they studies the jump portent experimentally and did numerical simulation to validate their result. They shows that the momentum flux is a major component of controlling and finding the jump position beyond this the variation ofjump thickness with stream rate of fluid is found from their surface work and numerical simulation.

Coomber et al. [5] did the experimental work to catch the relation between the stream speed and jump. In previous literature based on jump there was an assumption is made that the relation between the stream rate and radius of jump is linear but the found that the curve is non-linear and is logarithmic.

Brechet et al. [6] did both theoretical and experimental work and the problem associated with the radial jump by using elementary hydrodynamics and studied the law which governs the locality of circular jump and to correlate the prediction with investigational data. They plot with different flow rate and to find radius and either different spacing distance and deriver the relation for radius for viscous fluid.

Teamah et al. [7]. They explore the circular jump with jet disposition on a horizontal objective smooth plate. The profile formed was unique and non-circular shape owing to slanting jet inclination angle. The nozzle was inclined from (30° to 90°) and water stream rate from 2 lpm to 5 lpm with constant spacing between target plate to nozzle of 30 mm.

Katti et al. [8] study the portent of jump experimentally on the basis of Reynolds number and nozzle to plate arrangement and its effect on distribution of indigenous heat transfer owing to liquid jet which is submerged with air on target plate which is flat and smooth by using a jet which is obtained from cylindrical shaped nozzle having length to diameter ratio of 83.

Bohr's et al. [9] investigate experimentally the process of jump by using viscous fluid i.e. ethylene-glycol with water. Setup is made to control the layout amid the objective plate and jet. When the depth of is increased they find parting occur at the bottom with respect to transition state with broken wave.

Kavcic et al. [10] worked on two layer of fluid and investigated experimentally to present their result by keeping focus on stream rate, density difference in two fluid i.e. between different water and salinee water. When the stream rates are stable circular pattern formed which consist of3-4 well defined waves. This entire wave pattern deformed when the stream rates are higher. They maintain constant flow rate about three to four minute for stable circular profile after that instability appear when the flow rate increases which brakes this wave structure in turbulent motion.

Mikielewiczet al. [11]studide the jump portent formed owing to circular jet strike on flat surface. This effect is cause because of inertia force. The jump played into feature due to supercritical flow condition. They derived the expression for radius of jump by applying the principal of Bernoulli's equation, they did the preliminary analysis of the formation of 1st and 2nd type of jump and correlate their results with the experimental data.

Lienhard et al. [12]scruitnized experimentally and hypothetically on circular jet strike on his laboratory. He conferred the outcome of turbulence including stagnation zone and Nusselt number, heat transfer in jet strike which is laminar in nature are also discussed by himin his paper. He uses the jet for high heat flux cooling.

Liu et al. [13] suggested that the surface tension shows a vital part in the development of circular jump for jet strike. Surface tension effect the shape of jump. A series of uncertainties occur in the structure of jump if the effect of surface tension decrease than the flick thickness also decreases. This conclusion is verified by experiment on planar flick which cause exceptional jump structure.

3. OBJECTIVE

- Goal of the present work is to conduct a 2d axisymmetric model CFD analysis of water jet impinging on a convex target surface and measure the diameter of circular jump formed.
- To validate the data observed with the existing data on circular jump formed by impingement of water jet on straight plate.
- To analyse the outcome of curve radius of convexflick.
- To show the outcome of distance between the nozzle and the flick.

4. METHODOLOGY

CFD observation and analysis of circular jump on a convex target surface

The computational fluid dynamics analysis is carried out using ANSYS fluent for our objective study. The response strictures have been taken from the base paper. The prevailing equations such as continuity equation, momentum equation, energy equations, K equation and ε equations are used to perform this computational analysis.

Governing Equations

The equation for preservation of mass,

$$\frac{\partial\rho}{\partial t} + \nabla.(\rho\,\vec{v}) = S_m$$

Where S_m= mass included to the continuous phase or any handler sources.
For 2D axisymmetric model, the continuity equation is given by

$$\frac{\partial\rho}{\partial t} + \frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial r}(\rho v_r) + \frac{\rho v_r}{r} = S_m$$

Where x being the axial coordinate, r being the circular coordinate, v_x is the axial velocity, and v_r is the radial velocity.

Momentum Conservation Equations

Conservation of momentum in an inertial reference frame is described by

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F}$$

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{v} + \nabla\vec{v}^{\mathrm{T}}) - \frac{2}{3}\nabla.\vec{v}I\right]$$

Energy Equation

$$\frac{\partial}{\partial t}\sum_{k=1}^{n}(\alpha_k\rho_k E_k)+\nabla.\sum_{k=1}^{n}\left(\alpha_k\vec{v}_k(\rho_k E_k+p)\right) = \nabla.\left(k_{eff}\nabla T\right)+S_E$$

$k-\in$ model

The turbulence kinetic energy, k, and its rate of dissipation, ∈, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k v_i) = \frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\in -Y_M+S_k$$

and

$$\frac{\partial}{\partial t}(\rho\in)+\frac{\partial}{\partial x_i}(\rho\in v_i) = \frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\in}\right)\frac{\partial\in}{\partial x_j}\right]+C_{1\in}\frac{\in}{k}(G_k+C_{3\in}G_b)-C_{2\in}\rho\frac{\in^2}{k}+S_\in$$

Volume fraction equation

The tracing of the interface(s) between the phases is consummate by resolving the continuity equation for the volume fraction of one (or more) of the phases. For the qth phase (air in this study), this equation has the subsequent form (Walters and Wolgemuth 2009) (Fluent 2013):

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}(\alpha_q\rho_q)+\nabla.(\alpha_q\rho_q\vec{v}) = s_{\alpha_q}+\sum_{p=1}^{n}(\dot{m}-\dot{m})\right]$$

Where $\dot{m}$ is the mass transfer from phase q (air) to p (water) and $\dot{m}$ is the mass transfer from phase p (water) to phase q(air). The s_{α_q} is the source term and is the mass of phase q which is added to the continuous phase (if there is any) from the detached other phases (for example, owing to vaporization of liquid phase) and any handler sources. This source term is zero by default. $p_q$is the density ofthe q^{th} phase. In this study, there are

only two phases in all the simulations and it can be assumed that there will not be any mass transfer between the phases during the simulations. In further words, the other hand perspective of the equation (3.1) is equal to zero for the simulations of this study.

Cad model:

The cad model data for validation is booked from **Mohammad Khavari et al. [3]**with following description

Nozzle radius$\boldsymbol{R_N}$(mm)	5
Radius offlick (mm)	70
Height of nozzle from plate(mm)	10

Fig.4 CAD geometry of domain of straight plate

Meshing:

The meshing is done with a quad shaped structured grid size of 280X40 of 1 orthogonal quality using edge sizing and face meshing. The total elements produced in the meshing is 11200 and total nodes are 11521.

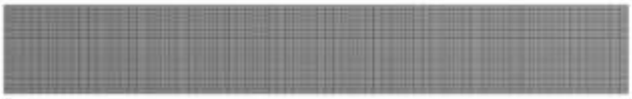


Fig. 5 meshing of domain

For makingflick convex, a convexing parameter is followed from **Ahmad Saberi et al. [2]**which is defined as

$$\eta = R_{TP}/R_C$$

Where η is adimensionless parameter, $R_C$curve radius of target plate (mm), R_{TP} radius of convex target surface (mm).

Curve radius of target plate is varied as 1000, 1300 ,1600 mm as per dimensionless parameter discussed in the previous paper at constant radius of convex target surface of 70 mm.

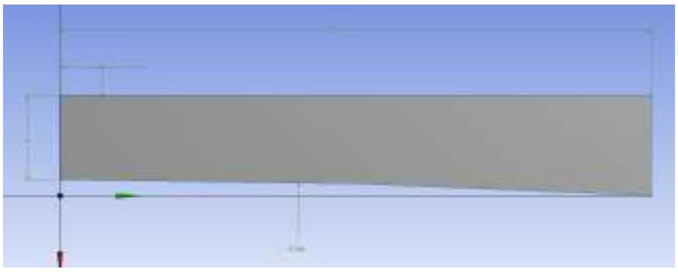


Fig.6 CAD geometry of domain of convex target surface

Meshing:

The meshing is done with a quad shaped elements by applying face sizing and face meshing. The total elements produced in the meshing is19140 and total nodes are 19544.

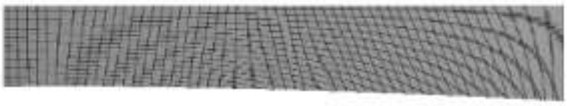


Fig.7 meshing of convex surface domain

Boundary conditions:

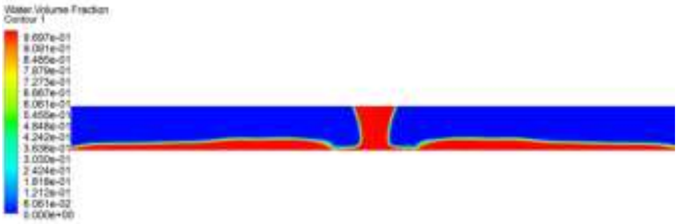
The inlet mass flow rate is varied from 10 ml per sec to 50 ml per second for straight plate and for convex target surface to demonstrate the influence of height of impingement the mass flow rate is retained constant at 30 ml per second. The upper boundary open to atmosphere is kept as pressure inlet boundary condition. Outlet is pressure outlet and target plate is modelled as adiabatic wall. Axis is made axisymmetric. A time dependent solver is castoff using implicit scheme. VOF model with implicit type formulation is used. Turbulence is modelled using k and epsilon equation with RNG model and near wall treatment is standard wall function. The working fluid is tap water with these properties:$\rho = 1000 kg/m^3, \nu = 1 \times 10^{-6} m^2/s$, and $\sigma = 0.073$ Nm. The radius of the incoming jet is $a = 5$ mm, the flow rate $Q = 30$ ml/s. Solution scheme for pressure velocity coupling is PISO. For spatial discretization of pressure PRESTO algorithm is used and second order upwind for momentum, compressive for volume fraction and second order upwind for turbulent kinetic energy and turbulent dissipation rate is used. Solution is initiated using standard initialization from inlet and two regions of thickness 2 mm is patched with water volume fraction of 1 at initial time. For calculating the solution the time step size is taken as 0.001 for 20 iteration per time step.

5. RESULTS AND DISCUSSION

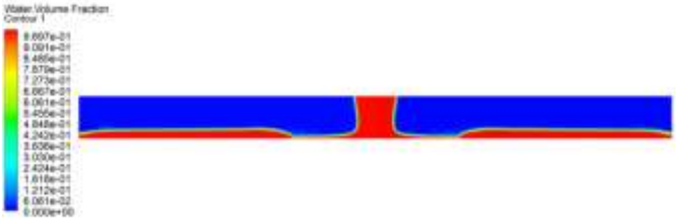
Mesh study

The mesh steadfastness was considered by a stricture called CPR defined as the figure of cells/ radius of the incoming jet. The jump replication for dissimilar standards of CPR for a water jet of 5 mm in radius striking a solid surface with astram rate of 30 ml/s and a downstream height of 2 mm. Handy review of the results displayed that for the instance with a CPR less than 10, no firm jump was moulded owing to a changing jump radius. When the CPR was enlarged to 15, the jump was steady with a radius of 26.28mm. Upsurging the CPR to 20 slightly changed the jump radius to26.38 mm. As a result, the jump radius does not alter significantly when upsurging the CPR value from 15 to 20. For all simulations performed in this study, therefore, a uniform mesh of20 cells per radius of the jet was used.[3]

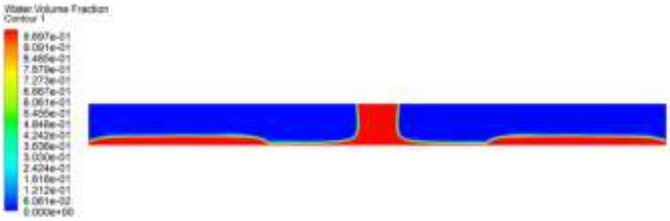
Contours of volume fraction of water impinging on flat target plate.



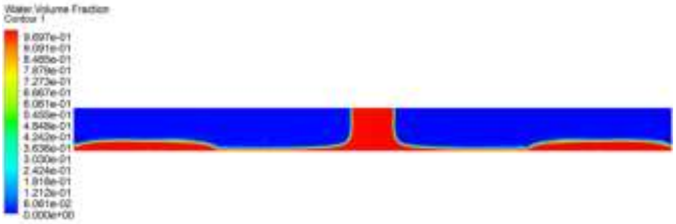
Q=10 ml/sec



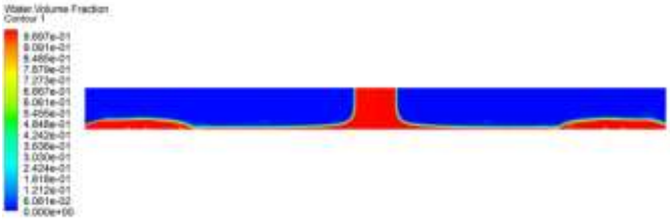
Q=20 ml/sec



Q=30 ml/sec



Q=40 ml/sec



Q=50 ml/sec

Fig. 8 volume fraction contours of water at different mass flow flow rate

Validation work:

For validation of the work results from Mohammad Khavari is compared with the present results as

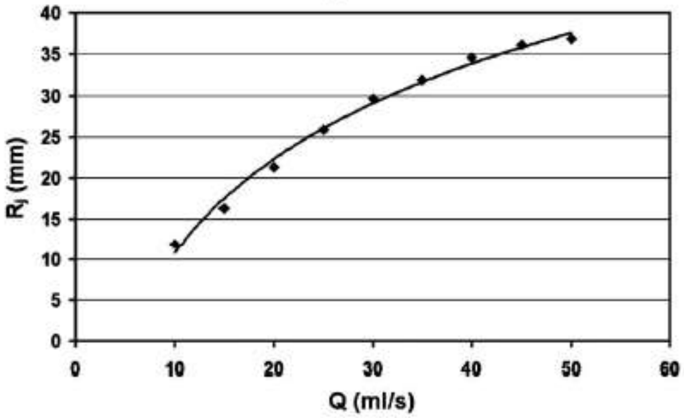


Fig. 9: Result from the previous work

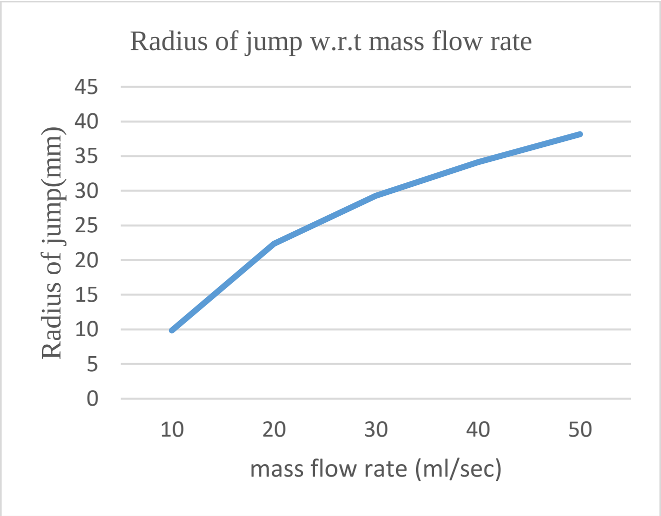
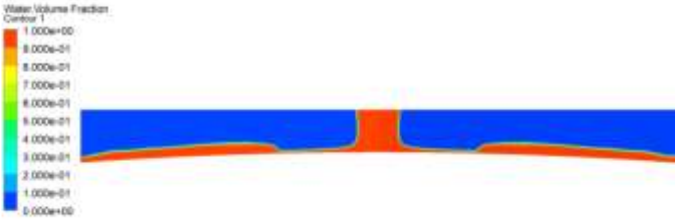


Fig. 10: Radius of jump w.r.t mass flow rate

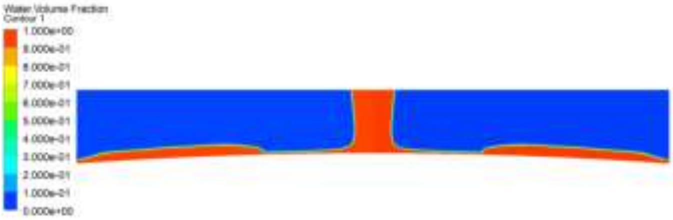
It is seen during the observation of the radius of circular jump the radius is being varied outwards. In the previous literature it is seen that the radius of circular hydraulicjump is some function of height of jump. For a stable calculation result the height is taken as 2 mm. The results at the constrained parameter matches with the existing result of formerly mentioned work. In the previous paper mass stream rate is varied between 10 ml/sec to 50 ml/sec and consequent radius of jump was observed to be varying between 11 mm to 37 mm in round off figure. In present work the curve radius varies between 9.84 mm to 38.16 mm with a maximum error of 9%.

CFD results for modified cases

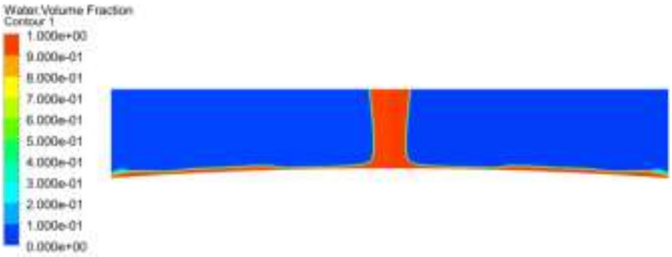
After performing the CFD calculation on modified cases which is the convex target surface impingement cases. The succeeding results were attained in contour form for visualization of volume fraction of water.



(a)



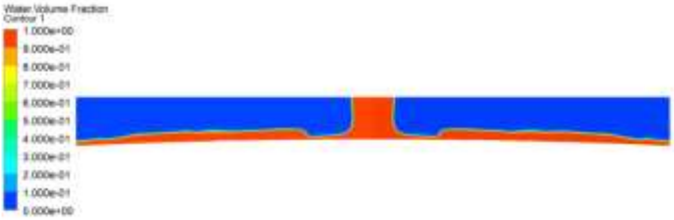
(b)



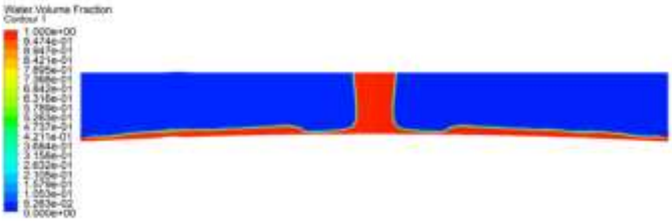
(c)

Fig contours of volume fraction at R_C=1000 and height of nozzle (a) 10 mm (b) 15 mm (c) 20 mm

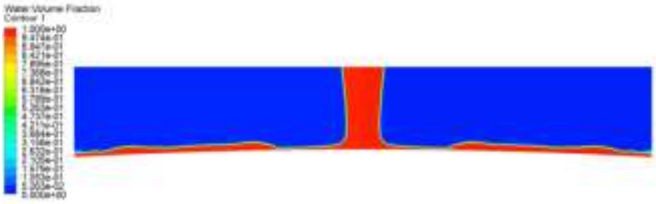
After estimating the numerical value radius of jump at constant curve radius of plate of 1000mm it is found that the values are 23.26 mm, 25.56 mm and 27.78 mm the nozzle height of 10,15 and 20 mm respectively .



(a)



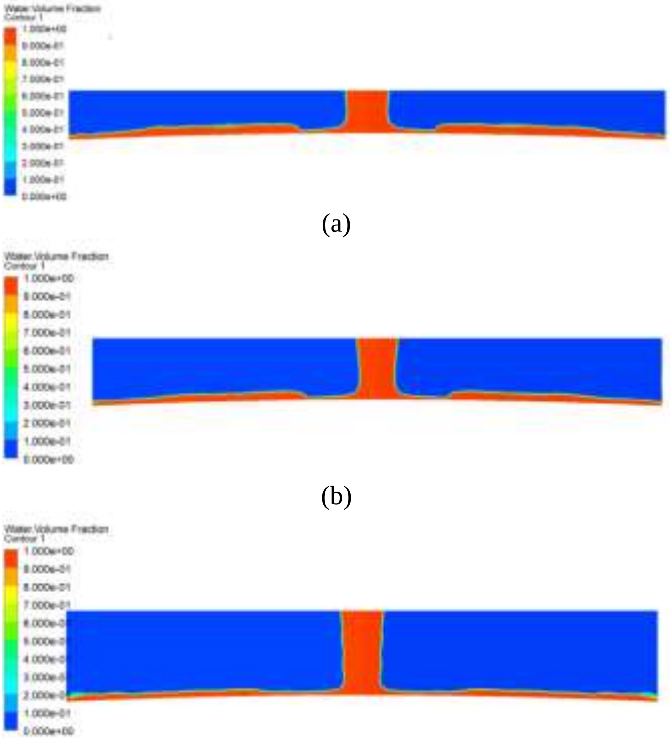
(b)



(c)

Fig.11: contours of volume fraction at R_C=1300 and height of nozzle (a) 10 mm (b) 15 mm (c) 20 mm

After estimating the numerical value radius of jump at constant curve radius of plate of 1300mm it is found that the values are 15.68 mm, 17.09 mm and 24.21 mm the nozzle height of 10,15 and 20 mm respectively .



(a)

(b)

(c)

Fig. 12: contours of volume fraction at R_C=1600 and height of nozzle (a) 10 mm (b) 15 mm (c) 20 mm

After estimating the numerical value radius of jump at constant curve radius of plate of 1600mm it is found that the values are 15.53 mm, 16.74 mm and 20.06 mm the nozzle height of 10,15 and 20 mm respectively .

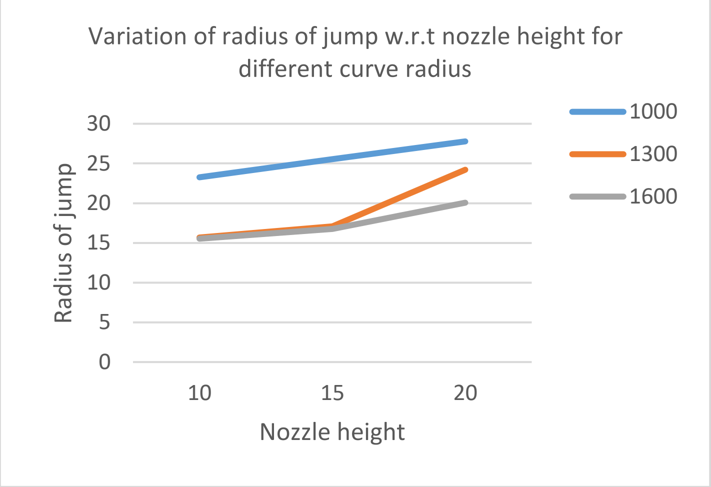


Fig.13 Variation of radius of jump w.r.t nozzle height for different curve radius

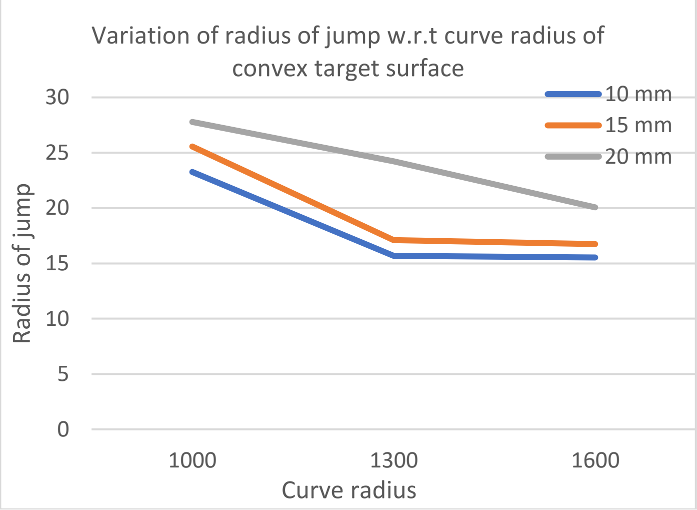


Fig.14 Variation of radius of jump w.r.t curve radius of convex target surface

6. CONCLUSION AND FUTURE SCOPE

After observing the data for jump radius w.r.t increase in nozzle height, it is found that at 1000 mm curve radius percentage upsurge in jump radius is 19.43%. For 1300 mm curve radius percentage increase is 54.42%. For 1600 mm curve radius the value is 29.16%. Also it is found that at constant height of 10 mm of nozzle with respect to increase in curve radius from 1000mm to 1600 mm the percentage decrease of 33.32%. After analyzing all the data of simulation it is concluded that increase in the height of nozzle from objective surface results in the increase of radius of circular jump which has further noteworthy role in heat transfer process. Also it is concluded that growing curve radius of target surface results in decrease in circular jump radius. The radius of circular jump varies as the height of downstream in the flow field. Although the circular jump radius is increased it is essential to study the effect on heat transfer due to curvature of convex target surface. An optimum height of nozzle can be calculated in future study for optimizing the compact space for regarding purpose.

REFERENCES

[1] Kate R. P., Das P. K., and Chakraborty S., 2007, "Jumps due to Oblique Impingement of Circular Liquid Jets on a Flat Horizontal Surface." Journal of Fluid Mechanics 573 pp. 247-263.

[2] Ahmad Saberi, Mohammad Reza Mahpaykar and A.R. eymourtash, Experimental Measurement of Radius of Circular Jumps: Effect of Radius of Convex target surface, Flow Measurement and Instrumentation,https://doi.org/10.1016/j.flowmeasinst.2019.01.011

[3] Mohammad Passandideh-Fard, Ali Reza Teymourtash, Mohammad Khavari, JOURNAL OF FLUIDS ENGINEERING, Vol. 133 / 011401-1

[4] Vishwanath K. P., Dasgupta R., Govindarajan R., and Sreenivas K. R., 2015, "The Effectof Initial Momentum Flux on the Circular Hydraulic Jump." Journal of Fluids Engineering 137(6), pp. 061301.

[5] Coomber P., Owen M., Harkness J., and Liam H., 2007,"Relationship between Flow Rates and the Radial Hydraulic Jump" GosfordHig School, Racecourse Road, Gosford NSW 2250.

[6] Brechet, Y., and Z. Néda., 1999, "On the circular Hydraulic Jump." American Journal of Physics 67(8), pp. 723-731.

[7] Teamah, M. A., Ibrahim, Dawood M. K. and Aleem M. M., 2012, "Experimental Investigation for Hydrodynamic Flow Due to Obliquely Free Circular Water Jet Impinging on Horizontal Flat Plate." European Journal of Scientific Research 83(1), pp. 60-75.

[8] Katti V. and Prabhu S. V., "Experimental Study and Theoretical Analysis of Local Heat Transfer Distribution between Smooth Flat Surface and Impinging Air Jet from a Circular Straight Pipe Nozzle." International Journal of Heat and Mass Transfer 51(17), pp. 4480-4495.

[9] Bohr T., Dimon P. and Putkaradze V., 1993, "Shallow-Water Approach to the Circular Hydraulic Jump." Journal of Fluid Mechanics 254, pp. 635-648.

[10] Kavčič, I., 2007, "An Experimental Study of the Circular Internal Hydraulic Jump."Geophysical Fluid Dynamics.

[11] Mikielewicz J. and Mikielewicz D., 2009, "A Simple Dissipation Model of Circular Hydraulic Jump." International Journal of Heat and Mass Transfer 52(1), pp. 17-21.

[12] Lienhard V., 2005 "Heat Transfer by Impingement of Circular Free Surface Liquid Jets."18th national & 7th ISHMT-ASME, Heat and Mass Transfer Conference January 4-6, IIT Guwahati India".

[13] Liu, X., 1993 "The Hydraulic Jump in Circular Jet Impingement and in other Thin Liquid Films." Experiments in Fluids 15(2), pp. 108-116.