ISME2010-3537

Experimental and Numerical Investigation of Circular Hydraulic Jump

Ali Reza Teymourtash[1], Mohammad Khavari[2] and Mohammad P-Fard[3]

[1] Assistant Professor, Mechanical Eng. Department, Ferdowsi University of Mashhad; teymourtash@um.ac.ir
[2] Graduate Student, Mechanical Eng. Department, Ferdowsi University of Mashhad; mohammad.khavari@yahoo.com
[3] Associate Professor, Mechanical Eng. Department, Ferdowsi University of Mashhad; mpfard@ferdowsi.um.ac.ir

Abstract

In this study, the circular hydraulic jump (CHJ) is investigated experimentally by using the setup constructed in the Fluid Mechanics Laboratory. Also VOF numerical method based on Youngs' algorithm is used to simulate the formation of the circular hydraulic jump. The results show that the numerical model is capable of simulating the jump formation and the numerical results are verified by the experimental observations. The both results indicate that enhancing the volumetric flow rate will increase the radius of the jump. Also, the agreement of both numerical and experimental results with the CHJ theory is satisfactory.

Keywords: Circular Hydraulic Jump, Experimental Setup, Numerical Simulation, Volume-of-Fluid Method.

1- Introduction

At the beginning of the twentieth century, the great British physicist, Lord Rayleigh encountered a discontinuity in the geometry of linear one dimensional flow. The structure is called *river bore* if moving, and *hydraulic jump*, if stationary and is created due to e.g. variation in river bed. The classical planar hydraulic jump which occurs in open-channel flows is a very old and well-known phenomenon thoroughly considered in the literature. However, the Circular Hydraulic Jump (CHJ) although having a similar name, is completely a different Phenomenon. When a circular vertical liquid jet impacts on a solid horizontal surface, which is called *target plate*, the flow spreads radially away everywhere – from the stagnation point – until at a particular radius, which is called the radius of the jump, the thickness of the liquid film increases abruptly and a so-called circular or axisymmetric hydraulic jump occurs.

As mentioned earlier, the first person who considered CHJ was probably Lord Rayleigh (1914) who proposed his model by using the continuity and momentum equations and assuming the flow as being inviscid [1]. He assumed that mass and momentum are conserved across the jump, but energy is not. He finally could derive some relations for the inviscid jump. Rayleigh's method was based on the analogy of shallow water and gas theories. The complete theory of inviscid circular hydraulic jump was presented by Birkhoff and Zarantonello in 1957 [2].

However, it is clear that the flow in such a problem is viscous and the inviscid theory is not adequate for predicting the location of the circular hydraulic jump occurrence, since the fluid layer thickness before the jump is typically sufficiently thin, so that the diffusion of vorticity from the lower boundary is dynamically significant. Therefore, the viscosity must be taken into account.

The first person, who considered the effect of viscosity in CHJ, was Watson in 1964 who solved the problem analytically. He, in a strong, long and highly-referred paper, described the flow in terms of a Blasius sublayer developing in the vicinity of the stagnation point, as on a flat plate, and also in terms of a similarity solution. By using the momentum equation, he could finally obtain some relations for predicting the radius of the jump. Watson's model will be considered in detail in the next section.

The validity of Watson's theory has been investigated experimentally by many different researchers throughout the world in the last four decades such as Watson himself [3], Olson and Turkdogan [4], Ishigai *et al.* [5], Nakoryakov [6], Bouhadef [7], Craik *et al.* [8], Errico [9], Vasista [10], Liu and Lienhard [11], Ellegaard *et al.* [12], and in particular Bush and Aristoff [13, 14]. The agreement between the theory and experiment has been very diverse, from good to bad, depending on the jump conditions. Even Watson himself has presented some data that are in poor agreement with his own theory.

Some other investigators also considered the problem from different aspects. Bowles and Smith studied the circular hydraulic jump -with surface tension considerations- and the small standing waves preceding the jump [15]. Higuera also proposed a model for planar jump by studying the flow in transition region in the limit of infinite Reynolds number [16]. Bohr *et al.* in 1993 obtained a scaling relation for the radius of the jump [17]. In 1997, they also proposed a simple viscous theory for free-surface flows that can accommodate regions of separated flow and yield the structure of stationary hydraulic jumps [18].

Watanabe *et al.* presented integral methods for shallow free-surface flows with separation in the application of circular hydraulic jump and also the flow down an inclined plane [19]. Ellegaard *et al.* who in 1996 investigated the CHJ empirically [12], for the very first time, observed the polygonal hydraulic jumps in their experiments [20] and reported them in detail in 1999 [21]. In the same year, Yokoi and Xiao considered the transition in the circular hydraulic jump numerically [22]. Three years later, they also studied numerically the structure formation in circular hydraulic jumps with moderate Reynolds numbers [23]. Brechet and Neda also investigated the circular hydraulic jumps and compared their theory and experiments [24].

Avedisian and Zhao studied in detail, the effect of gravity on the circular hydraulic jump and its different parameters experimentally [25]. Rao and Arakeri considered the CHJ empirically and measured the radius of the jump, film thickness and also the length of the transition zone and specially focused on jump formation and transition to turbulent flow [26]. In 2002, Ferreira *et al.* simulated the circular hydraulic jump numerically in order to compare the various upwind schemes for convective term of the Navier-Stokes equations [27].

Gradeck *et al.* studied the impingement of an axisymmetric jet on a moving surface both numerically and experimentally in order to simulate the cooling of a rolling process in the steel making industry [28].

As mentioned before, so far circular hydraulic jump has been studied numerically by other numerical methods and also for some other cases like an oblique jet. But in this study, the circular hydraulic jump is studied both experimentally and numerically and the method used for simulating the CHJ has been the method of Volume-of-Fluid based on Youngs' algorithm. The numerical results for the variation of jump radius with different flow rates are compared with the experimental measurements. Also the both results are compared with Modified Watson's theory.

2- Theory of Circular Hydraulic jump

Circular hydraulic jumps might take place, when a vertical descending liquid jet impacts a solid horizontal surface. Figure 1 shows a sample of an empirically observed CHJ in our experiments.

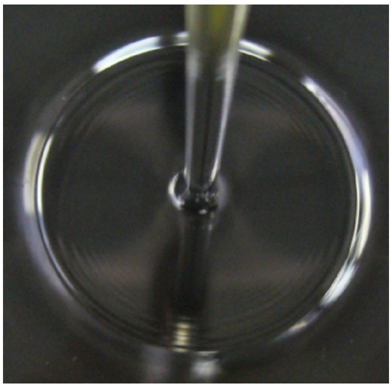


Figure 1: The circular hydraulic jump

The important feature of CHJ is its potential for heat loss in downstream of the jump, especially for the processes in which the purpose is cooling a hot surface [25]. The general structure of a circular hydraulic jump is shown in Figure 2.

Watson proposed two models for CHJ. His first model was an inviscid one for downstream of the jump in which he assumed the pressure force to be equal to the rate at which momentum is increasing. In his second model which was viscous, he had used the prandtl boundary layer theory for development of the flow which is considered here in brief.

Figure 2: The general structure of CHJ

In upstream region where the flow is viscous, Watson divided the flow field into four different regions: i.) The region very close to the stagnation point where the radial distance is of the same order of the jet radius $(r = O(a))$ and the boundary layer thickness is of order $d = O(ua/U_0)$ where a and U_0 are the radius and the velocity of the incoming jet and u is kinematic viscosity (see figure 2); ii) The region $r >> a$ in which the features of stagnation region are not important and the boundary layer is similar to the Blasius sublayer development over flat plate; iii) The region from the point where the boundary layer spans the whole fluid layer to the point where the velocity becomes self-similar that can be called a transition region; iv) The region in which the similarity solution suggested by Watson is valid.

According to Watson’s theory, the viscous solution is valid only in the second and fourth regions and for $\mathrm{Re} = Q/(ua) >> 1$ where $Q = pa^2 U_0$ is the volumetric flow rate. His approximate solution is clearly not correct in the first region, since the radii of the jump and the jet are of the same order. By neglecting the third region, Watson used the Karman-Pohlhausen method to match the solution of the second region (from Blasius velocity profile) and the solution of the last region for which he assumed the following velocity profile:

$$u = U(r) f\left(\frac{z}{d}\right) \tag{1}$$

where $U(r)$ is the velocity at the free surface and f is the similarity function.

By using the above velocity profile in momentum integral equation, Watson could derive this explicit relation for the thickness of the boundary layer:

$$r^2 d^2 - \frac{c^3\sqrt{3}}{p - c\sqrt{3}} \frac{ur^3}{U_0} = C \tag{2}$$

where $c = 1.402$ and C is the integration constant. By an order-of-magnitude analysis, Watson showed that in the region $r = O(a)$, $C = O(ua^3/U_0)$ and in the region $a << r < r_0$, $C = O(a^3/r^3)$ where $r_0 = 0.315 a \mathrm{Re}^{1/3}$ is the radial location in which the boundary layer absorbs the whole flow and is also shown by r_v in the literature. The proportionality factor for this critical radius, which is the place that the transition from the second region to the forth one is occurred, was obtained by matching the two different solutions just mentioned.

Watson ignored the integration constant C and obtained two relations for predicting the fluid layer depth x as:

$$x(r)=\frac{a^2}{2r}+\left[1-\left(\frac{2p}{3\sqrt{3}}c^2\right)\right]d \qquad r<r_0 \qquad (3)$$

$$x(r)=\frac{2p^2}{3\sqrt{3}}\frac{u(r^3+l^3)}{Qr} \qquad r\ge r_0 \qquad (4)$$

For reaching his main goal which was predicting the location of the jump occurrence, by assuming the downstream height to be known, Watson used the momentum balance and eventually could derive some relations for the jump radius. He also had ignored the effect of surface tension in his analysis.

Liu and Lienhard stated that if the radius of the jump decreases or radius of the jet increases, then the upstream Froude number will be larger. They concluded finally that for jumps with large downstream height and high upstream Froude number, the Watson's model is not accurate enough. Therefore, briefly it can be said that the accuracy of Watson's theory is not appropriate for jumps of small radius and height, known as weak jumps [11]. Based on the experiments, the surface tension influence is underscored in small jumps. The empirical observations have shown that reducing the surface tension causes the radius of the circular jump increase and also makes the jump more gradual, i.e. the jump becomes less abrupt.

Bush and Aristoff in 2003 have considered the influence of surface tension on CHJ analytically and could propose a very simple valuable relation for the curvature force –which for weak jumps is comparable with pressure forces in momentum equation- and eventually were capable of modifying Watson's theory, i.e. his relations for predicting the jump radius. These modified relations are [13]:

$$\frac{r_1 d^2 g a^2}{Q^2}\left(1+\frac{2}{Bo}\right)+\frac{a^2}{2p^2 r_1 d}= 0.10132-0.1297\left(\frac{r_1}{a}\right)^{3/2}\mathrm{Re}^{-1/2} \qquad r<r_0 \qquad (5)$$

$$\frac{r_1 d^2 g a^2}{Q^2}\left(1+\frac{2}{Bo}\right)+\frac{a^2}{2p^2 r_1 d}= 0.01676\left[\left(\frac{r_1}{a}\right)^{3}\mathrm{Re}^{-1}+0.1826\right]^{-1} \qquad r\ge r_0 \qquad (6)$$

where d is the downstream height (or outer depth which is also shown by h_∞), g is the gravitational acceleration, r_1 (also shown by r_j or R_j) is the radius of the jump, $Bo = rgR_j\Delta H / s$ is the Bond number and ΔH is the jump height. It must be mentioned that these relations are valid for laminar flow which is the flow regime considered here numerically. Watson also derived some similar relations for turbulent flow in CHJ problem, but they were hardly ever validated and confirmed by researchers, since the turbulent circular hydraulic jump is so complicated that it makes it very difficult to study it in detail.

The Bush and Aristoff relations for radius of the jump differ from those of Watson only in the term including Bond number that contains the surface tension effect which is highlighted in the weak jump regimes. By this modification to Watson's theory, Bush and Aristoff could improve the accuracy of his model in small jump regimes in which his own theory had some imperfections. According to the above relations, if the jump is big enough, then the Bond number will become large and its term in the equations becomes negligible and so the old Watson's relations will be obtained.

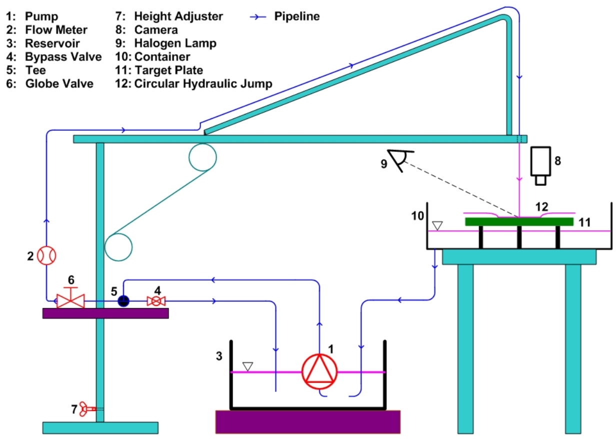
In 1993, Bohr *et al.* proposed a scaling relation for the circular hydraulic jump radius as:

$$R_j \sim q^{5/8}u^{-3/8}g^{-1/8} \qquad (7)$$

where $q = Q/(2p)$ and R_j is the radius of the jump. According to this relation, decreasing gravity and viscosity and also increasing the flow rate will result in bigger jumps. They verified the validity of this scaling relation by their own experimental observations.

3- Experimental Setup

The main purpose of doing the experiments was validating our own numerical code which was developed to be able to simulate the circular hydraulic jump. The experimental setup which is shown in Figure 3 was established in the Fluid Mechanics Laboratory at Ferdowsi University of Mashhad in order to study the circular hydraulic jump phenomenon empirically.



Figures 3: The schematic of the experimental setup

The experimental setup contains a reservoir, a pump, pipes of different diameters and some fixtures supporting them. The working fluid is pumped from the supply tank through the tubes and then enters the copper pipe which is long enough to make sure that the flow is laminar. The jet then leaves the pipe and impacts vertically on the circular disk and the jump is formed after which the liquid leaves the disk and fills a glass container which delivers the fluid to the reservoir again and the loop is now complete.

The internal diameter of the used pipe is $4.73mm$, the distance between the pipe exit and the disk is $5mm$, the diameter of the target plate over which the jump is formed is $30mm$ and the experiments are performed for several different flow rates. The working fluid is the tap water and the radius of the jump is obtained by digitizing the images taken from the jump by a digital camera. The downstream height is also measured by calipers and the flow rate is measured directly by using a calibrated tube and a timer. One of the samples of the jumps observed in our experiments was already shown in Figure 1.

4- Numerical Method

Also in this study, the circular hydraulic jump is simulated numerically by solving the Navier-Stokes equations, along with an equation for tracking the free-surface. In this section, we present a brief account of the numerical method. The governing equations are the continuity and momentum equations:

$$\nabla \cdot \mathbf{V} = 0 \tag{8}$$

$$\frac{\partial \mathbf{V}}{\partial t} + \nabla \cdot (\mathbf{V}\mathbf{V}) = -\frac{1}{\rho}\nabla p + \frac{1}{\rho}\nabla \cdot \tau + \mathbf{g} + \frac{1}{\rho}\mathbf{F}_b \tag{9}$$

where $\mathbf{V}$ is the velocity vector, p is the pressure, τ is the stress tensor and $\mathbf{F}_b$ represents the body forces acting on the fluid.

The free surface is tracked by using the volume-of-fluid (VOF) method by means of a scalar field f (known as volume of fluid fraction) whose value is unity in the liquid phase and zero in the vapor. When a cell is partially filled with liquid, i.e. the interface, f will have a value between zero and one:

$$f = \begin{cases} 1 & \text{in liquid} \\ >0, <1 & \text{at the liquid-gas interface} \\ 0 & \text{in gas} \end{cases} \tag{10}$$

The discontinuity in f is propagating through the computational domain according to:

$$\frac{df}{dt} = \frac{\partial f}{\partial t} + \vec{V} \cdot \nabla f = 0 \tag{11}$$

For the advection of volume fraction f based on Equation (11), different methods have been developed such as SLIC, Hirt-Nichols and Youngs' PLIC. The reported literature on the simulation of free-surface flows reveals that Hirt-Nichols method has been used by many researchers. In this study, however, we used Youngs' method [29], which is a more accurate technique. Assuming the initial distribution of f to be given, velocity and pressure are calculated in each time step by the following procedure. The f advection begins by defining an intermediate value of f :

$$\tilde{f} = f^n - dt\nabla \cdot (\vec{V} f^n) \tag{12}$$

Then it is completed with a "divergence correction":

$$f^{n+1} = \tilde{f} + dt(\nabla \cdot \vec{V}) f^n \tag{13}$$

A single set of equations is solved for both phases, therefore, density and viscosity of the mixture are calculated according to:

$$\rho = f\rho_l + (1-f)\rho_g \tag{14}$$

$$\mu = f\mu_l + (1-f)\mu_g \tag{15}$$

where subscripts l and g denote the liquid and gas, respectively. New velocity field is calculated according to the two-step time projection method as follows. First, an intermediate velocity is obtained:

$$\frac{\tilde{\vec{V}} - \vec{V}^n}{dt} = -\nabla \cdot (\vec{V}\vec{V})^n + \frac{1}{\rho^n}\nabla \cdot \vec{\tau}^n + \vec{g}^n + \frac{1}{\rho^n}\vec{F}_b^n \tag{16}$$

The continuum surface force (CSF) method is used to model surface tension as a body force ($\mathbf{F}_b$) that acts only on interfacial cells. Pressure Poisson equation is then solved to obtain the pressure field:

$$\nabla \cdot \left[\frac{1}{\rho^n}\nabla p^{n+1}\right] = \frac{\nabla \cdot \tilde{\vec{V}}}{dt} \tag{17}$$

Next, new time velocities are calculated by considering the pressure field implicitly:

$$\frac{\vec{V}^{n+1} - \tilde{\vec{V}}}{dt} = -\frac{1}{\rho^n}\nabla p^{n+1} \tag{18}$$

In this study, the numerical code for Volume-of-Fluid method has been developed to be capable of simulating the circular hydraulic jump and the results are validated by the experiments and the CHJ theory.

Mesh Study

The cell size used in this study was set based on a mesh refinement study in which the grid size was progressively increased until no significant changes were observed in the simulation results. The mesh resolution was characterized by a parameter called *CPR* defined as the number of cells per radius of the jet. Figure 4 shows the simulation of the jump for different values of *CPR* for a given case. According to this Figure, since the radius of the jump does not change significantly by increasing the *CPR* value from 15 to 20, so the optimum mesh size was found to be 20 cells per radius of the jet. This mesh size was used for all simulations throughout this paper.

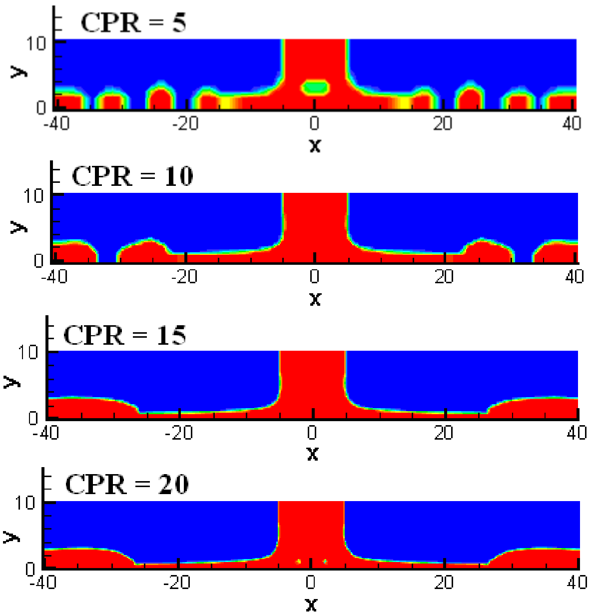


Figure 4: The simulation of the circular hydraulic jump for different *CPR* values

5- Results and Discussion

By using the Volume-of-Fluid method, the circular hydraulic jump is simulated and the results are verified by the experimental measurements. The location of jump formation and also how it occurs is seen in the numerical method.

Figure 5 shows the evolution of CHJ formation for a case selected from our experimental tests for which the flow rate is $Q = 11 ml/s$, the radius of the incoming jet before impact is $a = 1.8mm$ and the fluid is tap water

($r = 998 kg / m^3$, $u = 1.005 \times 10^{-6} m^2 / s$, $s = 0.073 N / m$). Note that the radius of the jet before impact is calculated using the Bernoulli equation by knowing the radius of the pipe exit and its height from the target plate, both measured experimentally, in order to improve the accuracy. The radius of the CHJ was measured (after image processing) to be $16.85 mm$, while the radius of the numerically simulated jump was $16.95 mm$ which shows a good agreement. The Reynolds No. of the jet in the experiments and also the simulations is in the range of 4000 < Re < 9000 which makes sure that the flow is laminar and the results might be compared with Watson's theory.

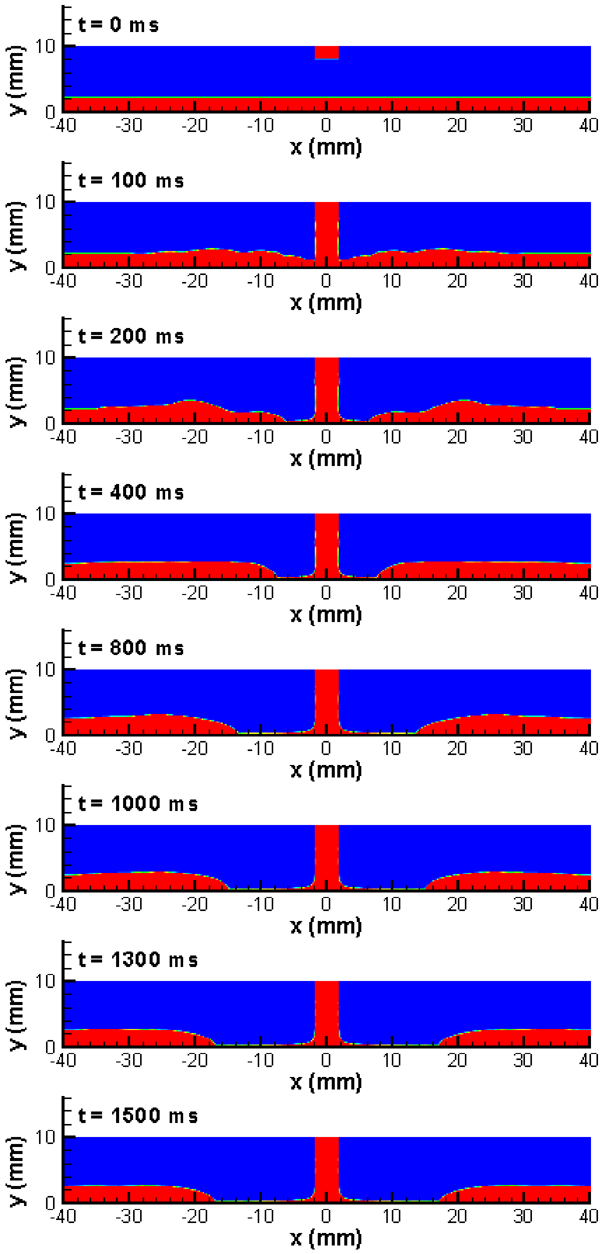


Figure 5: The evolution of formation of a circular hydraulic jump using VOF method for a given case

For other cases that were tested empirically, also a good agreement between the numerical and experimental jump radii is seen. Figure 6 shows the values of the CHJ radius from the numerical and experimental results for different flow rates.

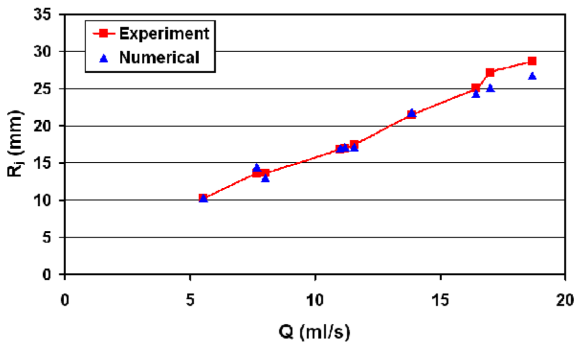


Figure 6: Comparison of jump radius from numerical and experimental results for different flow rates

We also compared both the numerical and experimental results with those of Modified Watson's theory (Equations 5 & 6) shown in Figure 7. It is seen that the agreement of these two with the theory is satisfactory.

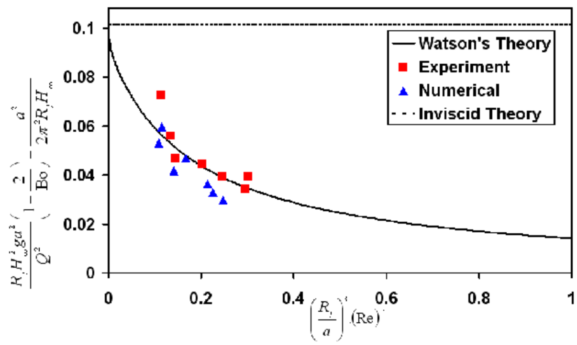


Figure 7: comparison of numerical and experimental results with those of Watson's theory

6- Conclusion

In this study, the impingement of a vertical liquid jet on a solid horizontal surface which leads to the formation of circular hydraulic jump was studied both experimentally and numerically by using the method of volume-of-fluid. The results show that this numerical method is capable of simulation the circular hydraulic jump. The numerical results are verified by the experimental measurements and the accuracy of the results for the values of jump radius is good. The results, both empirical and numerical, also show that the radius of the circular hydraulic jump increases by enhancing the volumetric flow rate. Also, the agreement of the numerical and empirical results with Modified Watson's theory is satisfactory. Also it is clearly seen that the inviscid theory is not adequate for predicting the radius of the jump.

7- Nomenclature

a	Jet Radius
Bo	Bond Number
d, h_∞	Downstream Height
f	Volume of Fluid Fraction
$\mathbf{F}_b$	Body Force
g	Gravitational Acceleration
p	Pressure

Q	Volumetric Flow Rate
R_j, r_j, r_1	Jump Radius
r_0, r_v	Critical Radius
Re	Reynolds Number
U_0	Incoming Jet Velocity
$\mathbf{V}$	Velocity Vector
t	Time
Greek Letters	
d	Boundary Layer Thickness
x	Fluid Layer Depth
u	Kinematic Viscosity
r	Density
s	Surface Tension
t	Stress Tensor

8- References

[1]- Rayleigh, Lord, 1914, "On the Theory of Long Waves and Bores", *Proc. R. Soc. Lond.* A 90, 324.

[2]- Birkhoff, G. & Zarantonello, E., 1957, *Jets, Wakes and Cavities,* Academic.

[3]- Watson, E. J., 1964 "The Radial Spread of a Liquid Jet Over a Horizontal Plane", *Journal of Fluid Mechanics*, 20, 481-499.

[4]- Olson, R. & Turkdogan, E., 1966, "Radial Spread of a Liquid Stream on a horizontal Plate", *Nature,* 211, 813-816.

[5]- Ishigai, S., Nakanishi, S., Mizuno, M. & Imamura, T., 1977, "Heat Transfer of the Impinging Round Water Jet in the Interference Zone of Film Flow Along the Wall", *Bull. JSME*, 20, 85-92.

[6]- Nakoryakov, V., Pokusaev, B., Troyan E., 1978, "Impingement of an Axisymmetric Liquid Jet on a Barrier", *International Journal of Heat and Mass Transfer*, 21, 1175-1184.

[7]- Bouhadef, M., 1978, "Etalement en Couche Mince d'un Jet Liquide Cylindrique Vertical Saur un Plan Horizontal", *Z. Anagew. Math. Physics*, 29, 157-167.

[8]- Craik, A., Latham, R., Fawkaes, M., Gibbon, P., 1981, "The Circular Hydraulic Jump", *Journal of Fluid Mechanics*, 112, 347-362.

[9]- Errico, M., 1986, "A Study of the Interaction of Liquid Jets with Solid Surfaces", PhD Thesis, University of California, San Diego.

[10]- Vasista, V., 1989, "Experimental Study of the Hydrodynamics of an Impinging Liquid Jet". B. Engineering Thesis, MIT.

[11]- Liu, X. & Lienhard, J., 1993, "The Hydraulic Jump in Circular Jet Impingement and in Other Thin Liquid Films", *Experiments in Fluids*, 15, 108-116.

[12]- Ellegaard, C., Hansen, A., Haaning, A., Hansen, K., & Bohr, T., 1996, "Experimental Results on Flow Separation and Transition in the Circular Hydraulic Jump", *Physica Scripta*, T67, 105-110.

[13]- Bush, J. W. M., Aristoff, J. M., 2003, "The Influence of Surface Tension on the Circular Hydraulic Jumps", *Journal of Fluid Mechanics*, 489, 229-238.

[14]- Bush, J. W. M., Aristoff, J. M., Hosoi, A., 2006, "An Experimental Investigation of the Stability of the Circular Hydraulic Jump", *Journal of Fluid Mechanics*, 558, 33-52.

[15]- Bowles, R. I., & Smith, F. T., 1992 "The Standing Hydraulic Jump: Theory, Computations and Comparisons with Experiments", *Journal of Fluid Mechanics*, 242, 145-168.

[16]- Higuera, F. G., 1994, "The Hydraulic Jump in a Viscous Laminar Flow", *Journal of Fluid Mechanics*, 274, 69.

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

[18]- Bohr, T., Putkaradze, V., Watanabe, S., 1997, "Averaging Theory for the Structure of Hydraulic Jumps and Separation in Laminar Free Surface Flows", *Phys. Rev. Lett*, 79, 1038-1042.

[19]- Watanabe, S., Putkaradze, V., Bohr, T., 2003, "Integral Methods for Shallow Free-Surface Flows with Separation", *Journal of Fluid Mechanics*, 480, 233-265.

[20]- Ellegaard, C., Hansen, A., Haaning, A., Hansen, K., Marcusson, A., Bohr, T., Lundbek Hansen, J., Watanabe, S., 1998, "Creating Corners in Kitchen Sinks", *Nature*, 392, 767.

[21]- Ellegaard, C., Hansen, A., Haaning, A., Hansen, K., Marcusson, A., Bohr, T., Lundbek Hansen, J., Watanabe, S., 1999, "Polygonal Hydraulic Jumps", *Nonlinearity*, 12, 1-7.

[22]- Yokoi, K., Xiao, F., 1999, "A Numerical Study of the Transition in the Circular Hydraulic Jumps", *Physics. Letters*, A 257, 153-157.

[23]- Yokoi, K., Xiao, F., 2002, "Mechanism of Structure Formation in Circular Hydraulic Jumps: Numerical Studies of Strongly Deformed Free-Surface Shallow Flows", *Physica*, D 161, 202-219.

[24]- Brechet, Y., Neda, Z., 1999, "On the Circular Hydraulic Jump", *American Journal of Physics*, 67 (8), 723-731.

[25]- Avedisian, C., Zhao, Z., 2000, "The Circular Hydraulic Jump in Low Gravity", *Proc. R. Soc. Lond*. A., 456, 2127-2151.

[26]- Rao, A., Arakeri, J., 2001, "Wave Structure in the Radial Film Flow with a Circular Hydraulic Jump", *Experiments in Fluids*, 31, 542-549.

[27]- Ferreira, V., Tome, M., Mangiavacchi, N., Castelo, A., Cuminato, J., Fortuna, A., McKee S., 2002, "High-order Upwinding and the Hydraulic Jump", *International Journal for Numerical Methods in Fluids*, 39, 549-583.

[28]- Gradeck, M., Kouachi, A., Dani, A., Arnoult, D., Borean, J., 2006, "Experimental and Numerical Study of the Hydraulic Jump of an Impinging Jet on a Moving Surface", *Experimental Thermal and Fluid Science*, 30, 193-201.

[29]- Youngs D. L., "Time Dependent Multi material Flow with Large Fluid Distortion" Numerical Methods for Fluid Dynamics, N.Y., 273-285, 1982.