International Journal of ChemTech Research
CODEN(USA): IJCRGG ISSN : 0974-4290
Vol.5, No.2, pp 1003-1008, April-June 2013

ICGSEE-2013[14th – 16th March 2013]
International Conference on Global Scenario in Environment and Energy

Numerical Modelling For Hydro Energy Convertor: Impulse Turbine

Vishal Gupta[1]*, Vishnu Prasad[2], Ruchi Khare[2]

[1]Department of Energy, M.A. National Institute of Technology, Bhopal, MP, India

[2] Department of Civil Engineering, M.A. National Institute of Technology, Bhopal, MP, India

*Corres.author: vishal.indomitable@gmail.com
Phone No: +91 755 405 1391, +91 8989717204

Abstract: The force by a free water jet on curved plate depends on the profile of the plate. The objective of this paper is to validate the results of numerical simulation of water jet on curved plates with different flow angles at outlet. Steady state numerical analysis based on two phase homogeneous model for free surface turbulent flow with standard k-ε turbulence model is done using ANSYS-CFX. The effect of plate profile on pressure, velocity and jet force has been studied. Comparison of theoretical and numerical force on curved plates has been made and found to have a close comparison.
Keywords- Computational fluid dynamics, free surface flow, multi-phase fluid flow, curved plates, jet force.

Introduction

When a nozzle is fitted at the outlet of a pipe, the water comes out in the form of the jet. This jet exerts impulse force on object placed in front of it and computed from Newton's second law of motion or from impulse momentum equation. The jet leaving the nozzle is surrounded by air. Due to fast growth in the field of computational capacity and numerical methods, computers are being used to analyze even very complex flow like multiphase free surface flows[7]. Detailed pressure plots and stream lines are obtained within specified flow domain which are either very difficult or not possible experimentally[6].

In present work, steady state numerical simulation for jet striking on symmetrical curved plates with 120°, 135°, 150°, 165°, 180° outlet angle have been covered using commercial code ANSYS CFX-13 to validate the numerical results with theoretical value. The pressure and velocity distribution pattern is also obtained to study the effect of plate profile on flow pattern. The continuity and momentum equations for turbulent flow are discretized using a high-resolution upwind scheme. The incoming jet is assumed to be ideal, with a constant velocity profile. The turbulence is taken into account using the *k*−ε turbulence model with the standard wall function[3]. The surface-tension force in the flow is also taken into account in simulation. The numerical results bear a close resemblance with the theoretical values.

Geometric Modeling And Boundary Conditions

The numerical flow simulation requires 2D or 3D geometry of flow domain depending on type of flow. The flow domain is divided into small elements forming mesh over which numerical method is applied for discretisation of governing flow equations.

Geometry

The geometry of jet with same cross sectional area and curved plate with different outlet angles placed normal to jet have been modelled using ANSYS ICEM CFD-13.0 software[2]. The flow domain consists of jet inlet, jet back, jet surface, plate front, plate and opening.

The assembled 3D view of jet and curved plate has been shown in fig.1.

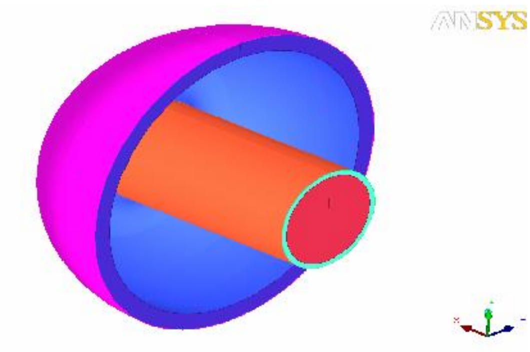

Fig 1: 3D view of jet with semi-spherical plate

Mesh Generation

The tetrahedral elements have been used for 3D flow domain and triangular elements for 2D surfaces. The meshing of domain for outlet angle of 180° has been shown in fig.2.

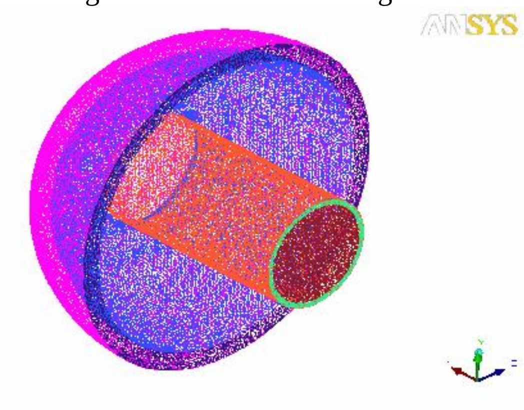

Fig 2: 3D view of mesh of semi-spherical plate.

The fluid flow region was discretized having 483360, 573300, 713268, 847513 and 975393 tetrahedral elements for plates with 120°, 135°, 150°, 165°, 180° outlet angles respectively.

Common Input Data

Air and water are taken as working fluids. Reference pressure is set to atmospheric. Buoyancy is activated and the value is given as -9.81. The Standard k-ε turbulence model with standard free surface model has been considered. The density of water is taken as 997 kg/m^3.

Boundary Conditions

Flow parameters like pressure, velocity or mass flow rate are given in the form of inlet and outlet boundary conditions to obtain numerical solution. The nature of the boundary condition affects the accuracy of numerical solution[4]. The following boundary conditions are used in present simulation.

Inlet boundary condition

This condition has been defined at inlet in the form of uniform water velocity as 50 m/s normal to surface. The volume fraction was given as 1 for water and 0 for air.

Wall condition

The plate is defined as smooth wall.

Outlet boundary conditions

As the jet flow is free surface and all boundaries except plate are defined as opening type boundary condition with reference pressure as 0 atmospheric.

Computation of Theoretical Force

The plate placed in front of jet reduces the jet velocity to zero in its direction and diverts the jet at an angle depending on geometry of the plate. As per Newton's Second Law of motion, the impulse force exerted by the jet on plate is calculated using the formula:

$$F_T = °AV_1(V_1 - V_2 \cos ¯) \qquad (1)$$

The energy losses due to friction and loss of mass is neglected in the above formula [1]. Hence actual force experienced by plate will be less than theoretical force given by the above equation. The ratio of computed and theoretical force is expressed as force coefficient[5] given below:

$$¸ = \frac{F_C}{F_T} \qquad (2)$$

The percentage deviation between theoretical and computed force is expressed as

$$¤ = \frac{F_T - F_C}{F_T} x100 \qquad (3)$$

Results And Discussions

The flow simulation has been carried out for constant cross-sectional area of the jet (1589 mm^2) and inlet jet velocity of 50 m/s. The distance between jet and plate (150mm) is also kept same in all cases. The root mean square (RMS) residual is set to 10^{-10} for the termination of iterations. The value of solver y+ is within acceptable limits (less than 200). The solver was run for 1000 iterations. The simulation has provided numerical impulse force, stream lines, pressure distribution etc. within the flow domain.

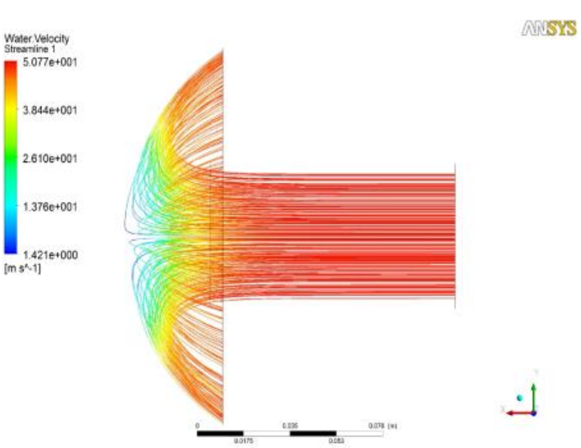

Fig 3: Stream lines for 150°curved plate

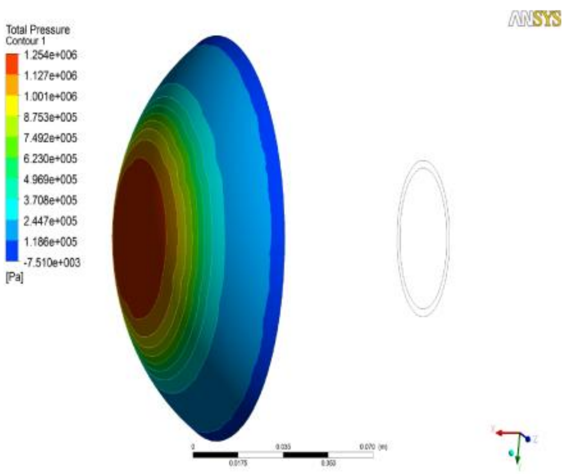

Fig 4: Pressure contour for 150°curved plate

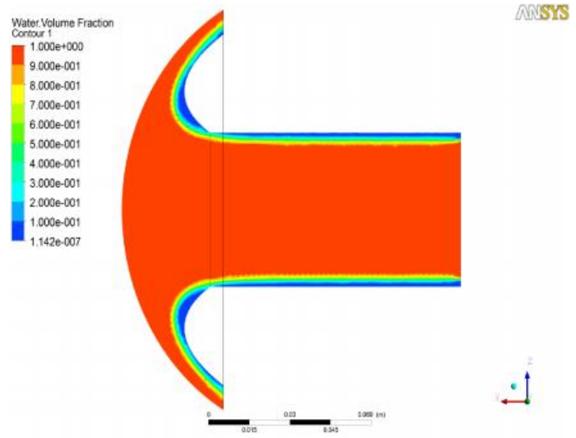

Fig 5: Water volume fraction contour on Z-X plane for 150°curved plate.

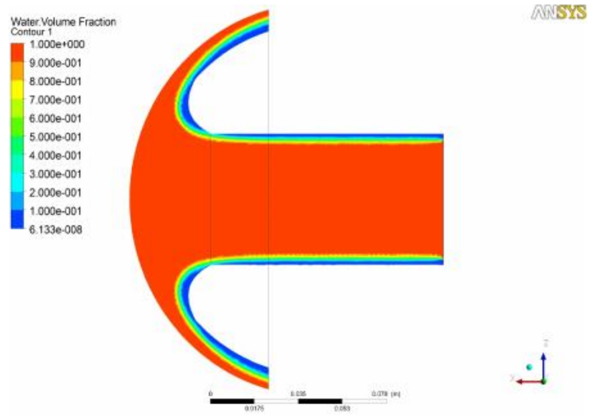

Fig 8: Water volume fraction contour on Z-X plane for 165°curved plate.

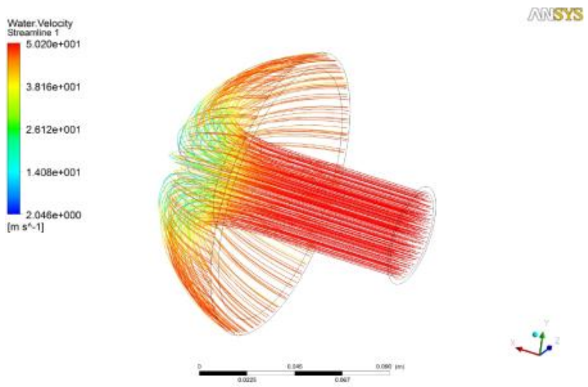

Fig 6: Streamlines for 165°curved plate

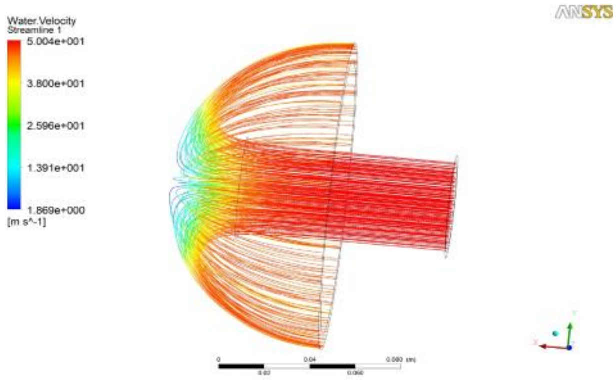

Fig 9: Streamlines for 180°curved plate

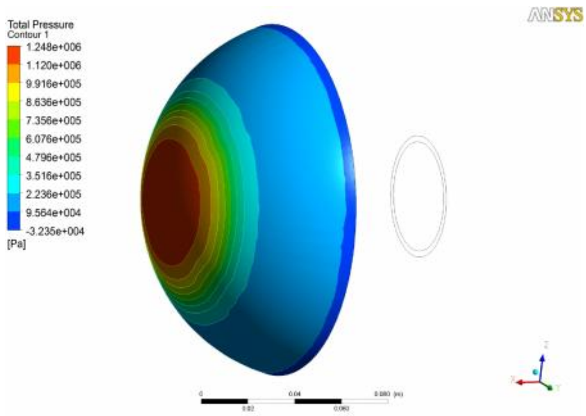

Fig 7: Pressure contour for 165°curved plate

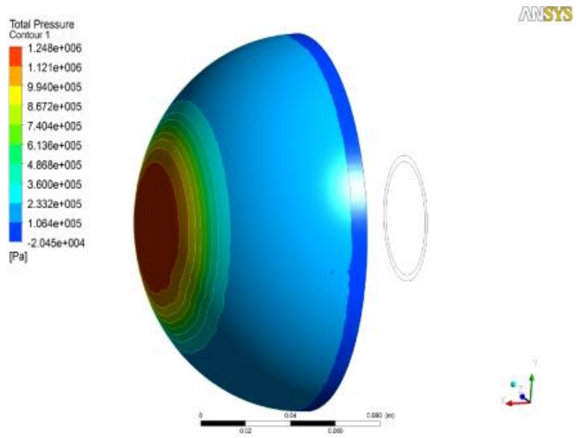

Fig 10: Pressure contour for 180°curved plate

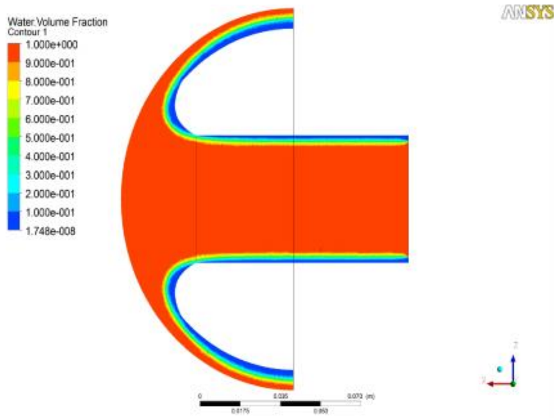

Fig 11: Water volume fraction contour on Z-X plane for 180°curved plate

The streamlines in jet are parallel before striking the plate and spreads after strike of jet to the plate. As the outlet angle of the plate increases, the thickness of the water sheet decreases due to flow over the larger area of the plate before exit. Also this leads to reduction in water jet velocity at the outlet. The velocity of water leaving the plates is 48.16 m/s, 47.50 m/s, 47.47 m/s, 46.98 m/s, 46.44 m/sec for plates with 120°, 135°, 150°, 165° and 180° outlet angles respectively.

The pressure at the mid of the plate is maximum as the velocity of the jet while flowing over plate is least. The momentum of water changes due to deflection of jet and this leads to force by jet on plate.

Iso-surfaces for water volume fraction (WVF), which is the ratio of volume of water to the total volume of water and air are also obtained. The water volume fraction 1 refers to the condition where there is 100% water and water volume fraction 0 refers to the condition where there is 100% air. The water volume fraction contours on X-Z iso-surface are shown in figure 5, figure 8, figure 11. The thickness of the water sheet reduces with the increase in outlet angle of the plate.

Table No. 1: Comparison of results

PLATE PROFILE	F_T (N)	F_C (N)	ζ	Φ
Outlet angle 120°	5946.22	5615.67	0.944	5.56
Outlet angle 135°	6737.22	6486.38	0.963	3.73
Outlet angle 150°	7397.20	7032.40	0.951	4.93
Outlet angle 165°	7793.22	7450.17	0.956	4.40
Outlet angle 180°	7928.30	7592.81	0.958	4.23

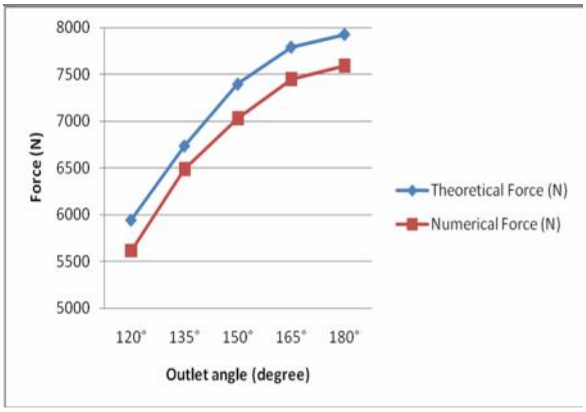

Fig 12: Graph showing variation of forces with different outlet angle of plates

The comparison between the theoretical and numerically computed forces is given in Table 1. It is observed that the deviation between theoretical and numerical computed force decreases with the increase of outlet angle. Both theoretical and computed force increases as the outlet angle increases which validate the numerical simulation. The rate of increase of force with outlet angle is more at the low angles and become nearly constant after angle 165°.

The difference between the numerical and theoretical results can be explained as in theoretical calculations, the jet is considered ideal and surface tension is not taken into account. In present case CFD analysis is done by taking care of the surface tension force also.

Conclusion

The numerically computed force bears close resemblance to the theoretical force. The maximum pressure experienced by the curved plate is approximately same for all the cases and lies at the centre of the plate. This is due to the reason that the inlet velocity of jet striking the plates is same for all the cases and its velocity reduces to zero at the centre of the plate[6]. The comparison of theoretical and numerical forces leads to the conclusion that similar methodology can also be applied for the performing the simulation of impulse turbines.

Nomenclature

A Cross sectional area of jet (m^2)
V_1 Velocity of jet (m/s)
V_2 Velocity of water at plate outlet (m/s)
F_T Theoretically computed force (N)
F_C Numerically computed force (N)
ζ Ratio of numerically computed force to theoretically computed force
Φ Percentage deviation ofnumerically computed force to theoretically computed force
ρ Density of water (Kg/m^3)
θ Angle between jet velocities at inlet and outlet (^{o})

References

1. Nechleba Miroslav “Hydraulic Turbines: Their design and equipment”, Publisher: Artia, 1957
2. Etienne Parkinson “New Developments in CFD Extend Application to Pelton Turbines”, CFX Update, 2003.
3. Perrig Alexandre, Avellan François, Kueny Jean-Louis, Farhat Mohamed “Flow in a Pelton Turbine Bucket: Numerical and Experimental Investigations” Transaction of ASME, Journal of Fluid Engineering, Vol. 128, pp. 350-358, 2006.
4. Zoppe B., Pellore C, Maitre T., Leroy P. “Flow Analysis Inside a Pelton Turbine Bucket” Transaction of ASME, Journal of Turbomachinery, Vol. 128, pp. 500-511, 2006.
5. Konnur M.S., Patel Kiran “Numerical Analysis of Water Jet on Flat Plate” Proceedings of 33rd National Conference on Fluid Mechanics and Fluid Power. Raipur. India, 2006.
6. Patel K, Patel B, Yadav M, and Foggia T “Development of Pelton turbine using numerical simulation” IOP Conf. Series: Earth and Environmental Science 12, 2010.
7. Gupta Vishal, Prasad Vishnu, “Numerical Computation of Force for Different Shapes of Jet Using CFD”, Proceedings of 38th National Conference of Fluid Mechanics and Fluid Power, MANIT, Bhopal, 2011, pp CFD-18.
8. ANSYS CFX-13 software manuals.

*****