International Association of Scientific Innovation and Research (IASIR)
(An Association Unifying the Sciences, Engineering, and Applied Research)

ISSN (Print): 2279-0047
ISSN (Online): 2279-0055

International Journal of Emerging Technologies in Computational and Applied Sciences (IJETCAS)

www.iasir.net

Analysis of Incompressible fluid flow over wedge with different angles

Dr. Deepak Sharma[1], Mr. Swapnil Jain[2], Ankush Kumar[3]
[1,2,3]Department of Mechanical Engineering
[1]Shri Shankaracharya Institute of Engineering and Technology,[2,3] Shri Shankaracharya Engineering College
Bhilai, Chhattisgarh, INDIA

Abstract: *The title of our research paper is "Analysis of Incompressible fluid flow over wedge with different angles". The basic objective of this research work is to optimize the fluid flow over different wedges and it angles. The optimization is done by varying wedge angle, its parameters considering fluid flow parameters also. The analysis of wedge is carried out in both 3D and 2D from which various results and graph plots are obtained. By looking to the results we can know the best wedge parameters to be used according to its application.*

Keywords: *incompressible; wedge; optimization; analysis*

I. Introduction

Fluid Flow

Motion of a fluid subjected to unbalanced forces or stresses. The motion continues as long as unbalanced forces are applied. A fluid may be a liquid, vapor, or gas. The term vapor denotes a gaseous substance interacting with its own liquid phase, for example, steam above water. If this phase interaction is not important, the vapor is simply termed a gas. The physical properties of a fluid are essential to formulating theories and developing designs for fluid flow. Especially important are pressure, density, and temperature. Since shear stresses cause motion in a fluid and result in differences in normal stresses at a point, it follows that a fluid at rest must have zero shear and uniform pressure at a point. This condition is known as hydrostatic condition.

II. Literature Review

Chen and Kadambi (1994) have conducted experiments in a vertical pipe. In their study, pressure gradients have been measured for vertical flow of slurry of particles with diameters between 1.37 mm and 3.4 mm in pipes of diameter 26 mm and 40 mm.

Doron and Barnea (1996) presented different flow patterns that are observed in horizontal liquid-solid pipeline transportation.. One of the important correlations on deposition velocity was given by Durand (1963) considering the diameter of the pipe (D), density of solid (ρs), density of liquid (ρl), and Froude number (FL). The correlation is given as:

$$V_D = F_L\sqrt{\left[2gD\left(\frac{\rho s-\rho l}{\rho l}\right)\right]}$$

This correlation does not consider the particle diameter (d) and initial solids concentration (Cv). Wasp et al. (1970) modified Durand's correlation and derived a deposition velocity equation for lower concentrations which is given as:

$$V_D = 1.87\left(\frac{d}{D}\right)^{1/6}\sqrt{\left[2gD\left(\frac{\rho s-\rho l}{\rho l}\right)\right]}$$

The above correlation describes the behavior of 1% dilute suspension. Later Wasp et al. (1977) derived a correlation describing the behavior of higher concentration slurries (up to 40%). It is given as:

$$V_D = 4.0\left(\frac{d}{D}\right)^{1/6}(C_v)^{1/5}\sqrt{\left[2gD\left(\frac{\rho s-\rho l}{\rho l}\right)\right]}$$

The above three correlations are the most important correlations for deposition velocity.

Achim Daniela et al (1999) have studied the experimental and computational data of tube erosion in a fluidized bed and simulated the work with CFX code having computational model of hydrodynamics and finnie model of erosion.

Manickam M et al (1999) have modeled the flow in the bifurcation duct of a power generation boiler plant. The computational fluid dynamics code was customized to determine erosion rate cause by particles that hit the duct wall. .

P.S.V.S. Sridhar et al (2003) have studied dilute solid-fluid flow over a wedge in a stationary channel which is numerically solved using one-way coupling between fluid and solid particles.

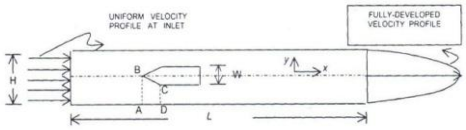


Figure 1: Flow domain-wedge placed along the channel axis

III. Problem Identification

This impact wear is mainly occurred in the tongue area of a centrifugal pump. And the bends in the pipe flow where the velocity of fluid is very high.

Impact wear is also occurred at the curved vanes of centrifugal pump those are handing slurries of variable densities and heterogeneous fluid.

Proposed work plan (Problem Formation):

i) Proposed research work is to prepare the CFD modeling for calculation of impact wear co-efficient [EI(α)] for dense slurry and multi size particle.

ii) By calculation of [EI(α)] in CFD modeling for different size particle and in multiphase flow. We can predict the approx. accurate time of pipes, curved vanes, open channel bends for replacement.

Proposed experimental set-up:

a) Computation of two phase flow field within the component.

b) Correlation of local flow conditions (velocity and concentration) near the wear surface to the local wear rate via a suitable wear model.

c) Empirical determination of wear co-efficient.

d) Erosion wear mainly occurred due to sliding wear and impact wear.

e) Impact wear rate is correlated to the K.E. flux of the particles as:

$$\mathbf{W_I} = \frac{\rho s C s V s^3}{EI(\alpha)}$$

Where ρs is the solids density
Cs is particle concentration
Vs is particle impact velocity
EI (α) is the impact wear coefficient as a function of the impact angle α.
Note that ρ*sCsVs* is the mass flux.

For prediction of impact wear co-efficient of test rig shown in figure 4, in this figure there is open channel through which solid-liquid flow strikes the impact wear specimen and causes erosion wear. Solid liquid flow through a channel strikes the impact wear specimen & causes erosion of wear sample. By measuring the wear depth for a given test duration the impact wear Specific energy co-efficient for a given slurry/wear material combination is calculated.

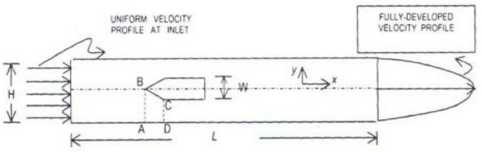


Figure 2

In this research paper we will focus our effort on calculating velocity Vs and the velocity profile of the flow and we are doing this research with water as the fluid.

A. Mathematical Modelling

1) Newton's Second Law

a) For a solid mass: F = m.a

b) For a continuum: $\rho\left[\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}.\nabla\mathbf{u}\right] = \nabla.\sigma + f$ Where ρ is mass per volume(Density)

c) Expressed in terms of velocity field u(x,y,z,t). In this form the momentum equation is also called Cauchy's law of motion.

d) For an incompressible Newtonian fluid, this becomes:

$$\rho\left[\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}.\nabla\mathbf{u}\right] = -\nabla p + \mu\nabla^2\mathbf{u}$$

2) Navier-Stokes and Bernoulli equations

a) When:

The flow is steady: $\partial u / \partial t = 0$

The flow is irrotational: the vorticity $\omega = \nabla \times \mathbf{u} = 0$

The flow is inviscid: $\mu = 0$

b) and using: $\mathbf{u}.\nabla\mathbf{u} = \frac{1}{2}\nabla(\mathbf{u}.\mathbf{u}) - \mathbf{u} \times \nabla \times \mathbf{u}$

c) We can rewrite the Navier-Stokes equation:

$$\rho\left[\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}.\nabla\mathbf{u}\right] = -\nabla p + \mu\nabla^2\mathbf{u}$$

as the Bernoulli equation: $\nabla\left(\frac{p}{\rho} + \frac{\mathbf{u}.\mathbf{u}}{2}\right) = 0$

1) k - ϵ model

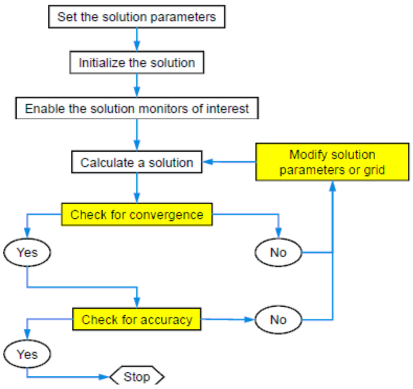
At high Reynolds numbers the rate of dissipation of kinetic energy is equal to the viscosity multiplied by the fluctuating vorticity. An exact transport equation for the fluctuating vorticity, and thus the dissipation rate, can be derived from the Navier Stokes equation. The k - epsilon model consists of the turbulent kinetic energy equation and the dissipation rate equation

$$\frac{\partial k}{\partial t} + \mathrm{div}(\rho \underline{u} k) = \mathrm{div}\left(\left[\mu_{lam} + \frac{\rho\nu_t}{\sigma_k}\right]\mathrm{grad}k\right) + \rho\nu_t G - \rho\epsilon$$

$$\frac{\partial \epsilon}{\partial t} + \mathrm{div}(\rho\underline{u}\epsilon) = \mathrm{div}\left(\left[\mu_{lam}+\frac{\rho\nu_t}{\sigma_\epsilon}\right]\mathrm{grad}\epsilon\right) + C_{1\epsilon}\rho\nu_t G\frac{\epsilon}{k} - C_{2\epsilon}\rho\frac{\epsilon^2}{k}$$

B. Setup & solution

FLUENT requires inputs (solver settings) which tell it how to calculate the solution. By introducing the boundary condition we can command the software to calculate in its limits. Emphasis will be placed on convergence, which is critical for the CFD simulation. The sketch below shows the basic workflow for any simulation.



V. Results

The Following results were obtained after performing the analysis under ANSYS FLUENT 14.0 when the wedges were constructed at different nose angle and we have also seen the effect when the wedges were made blunt with a nose radius of 0.25 mm. The data's obtained are as follows:

2-D WEDGE WITH DIFFERENT NOSE ANGLES:

The above shown analysis was also performed under 2-D within the same circumstances and different nose angle such as 30^0, 45^0, 60^0. The graphs obtained are:

2-D WEDGE WITH NOSE ANGLE OF 30^0:

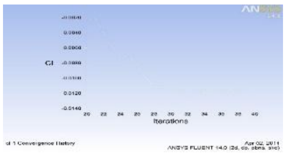


Figure 3. Coefficient of lift

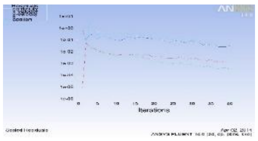


Figure 4. Scaled Residuals

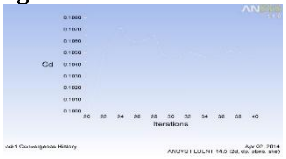


Figure 5. Coefficient of drag

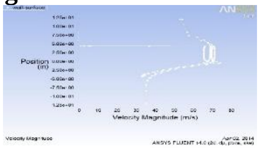


Figure 6. Velocity profile

2-D WEDGE WITH NOSE ANGLE OF 45^0:

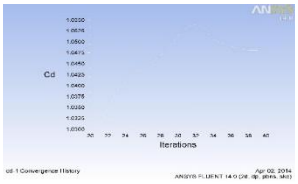


Figure 7. Coefficient of drag

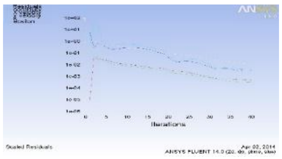


Figure 9. Scaled Residuals

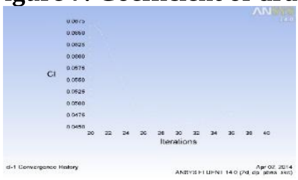


Figure 8. Coefficient of lift

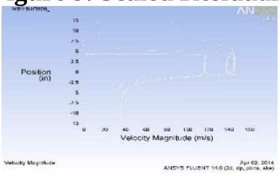


Figure 10. Velocity profile

2-D WEDGE WITH NOSE ANGLE OF 60^0:

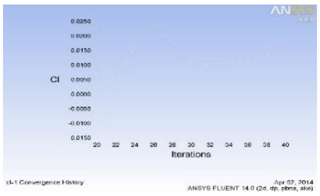


Figure 11. Coefficient of lift

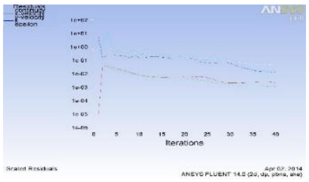


Figure 12. Scaled Residuals

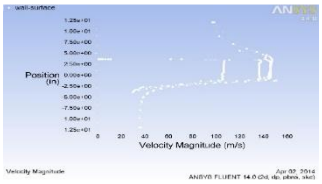


Figure 13. Velocity profile

2-D WEDGE WITH NOSE ANGLE OF 30^{0}AND BLUNT WITH NOSE RADIUS OF 0.25 mm:

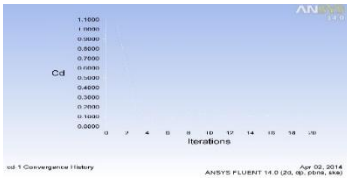


Figure 14. Coefficient of drag

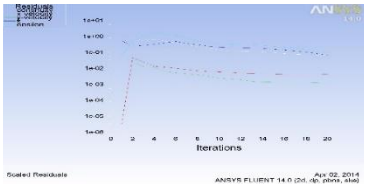


Figure 15. Scaled Residuals

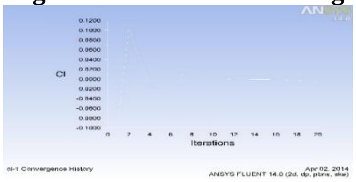


Figure 16. Coefficient of lift

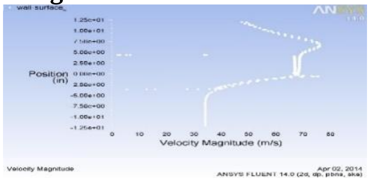


Figure 17. Velocity profile

2-D WEDGE WITH NOSE ANGLE OF 60^0 AND BLUNT WITH NOSE RADIUS OF 0.25 mm:

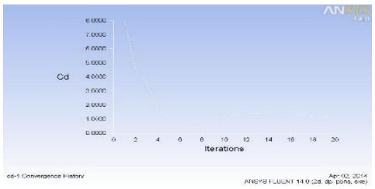


Figure 18. Coefficient of drag

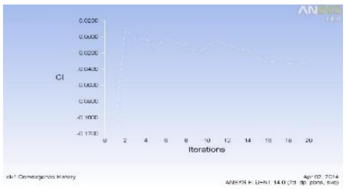


Figure 19. Coefficient of lift

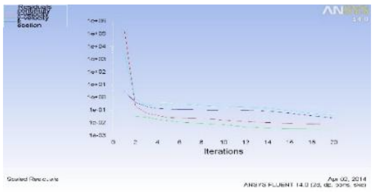


Figure 20. Scaled Residuals

VI. Conclusion

At same velocity inlet and pressure outlet the inlet pressure is least at angle of 45^0 and inlet pressure increases with decrease and increase in angle.

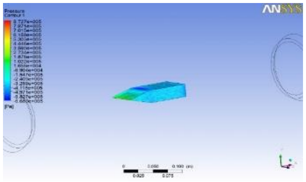


Figure 21. 30 degree pressure

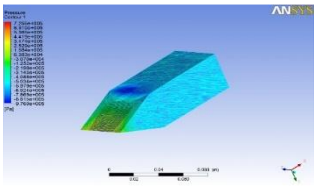


Figure 22. 45 degree pressure

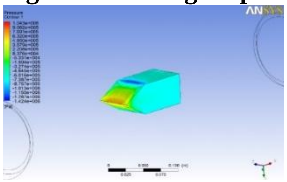


Figure 23. degree pressure

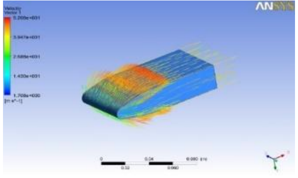


Figure 24. velocity vector 30 degree

At sharp edges we get more pressure and eddies are forming and these losses can be minimized by making the edge blunt. Velocity at outlet increases with increase in wedge angle.

At sharp edges we get more pressure and eddies are forming and these losses can be minimized by making the edge blunt. Velocity at outlet increases with increase in wedge angle.

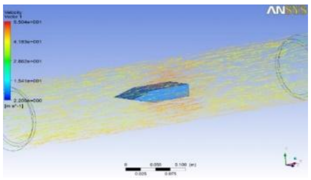


Figure 25: velocity 30 degree

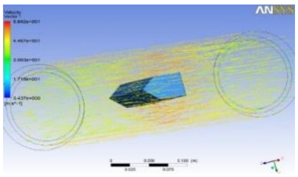


Figure 26: 45 degree velocity

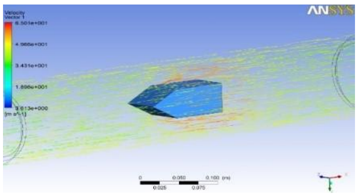


Figure 27: 60 Degree velocity

VII. Future Scope

The Future works that can be implied through this research paper are as follows:

A) The fundamental equation for calculation of impact wear rate is

$$W_I = \frac{\rho s C s V s^3}{E I(\alpha)}$$

For calculation of wear in any object requires the quantity $EI(\alpha)$.

B) We have carried out our simulation in a water medium; the experimental analysis for slurry having multiple density particles needs to be done.

C) The proposed experimental setup needs to be completed for the validation of our work.

References

[1] Sridhar P.S.V.S., Pagalthivarthi Krishnan V. and SanghiSanjeev. 2003. *Simulation of Particle Impact with a Wedge in Dilute Two-phase Flow*. Defence Science Journal. 53(2):199-204

[2] Pagalthivarthi K.V. and Gupta P.K.2009. Prediction of ErosionWear in Multi-Size Particulate Flow through a Rotating Channel. Tech Science Press FDMP.5(1): 93-121

[3] PagalthivarthiK.V.,Gupta P.K.,TyagiVipin and Ravi M.R.2011. *CFD Predictions of Dense Slurry Flow in Centrifugal Pump Casings.* World Academy of Science, Engineering and Technology.51

[4] Patnika Amar, SatapathyAlok, chandNavin and Barkoula N.M. 2010.Solid particle erosion wear characteristics of fiber and particulate filled polymer composites: A Review.Elsevier.Wear 268: 249–263

[5] PatilMilind S.,DeoreE.R.,Jahagirdar R.S. and Patil S.V. 2011.*Study of the parameters Affecting Erosion Wear of Ductile Material in Solid-Liquid Mixture*. World Congress on Engineering. 3: ISSN:2078-0958(Print)

[6] Kim Changhee, Lee Mansoo and Han Cheolheui 2008. *Hydraulic Transport of Sand Water Mixture in Pipelines*. Journal of Mechanical Science and Technology.22 :2534-2541

[7] Yuliang Zhang, Baoling CUI and Huashu 2013. *Numerical Simulation and Analysis of Solid liquid Two phase Flow in Centrifugal Pump*. Chinese Journal of Mechnical Engineering.26

[8] Clark H. McI. 2004.*The influence of the squeeze film in slurry erosion*. Journal of wear.256: 918-926.

[9] Gandhi B.K. and BorseSatish V. 2004. Nominal particle size of multi- sized particulate slurries for evaluation of erosion wear and effect of fine particles. Journal of wear. 257: 73-79.

[10] Tian Harry H., Graeme R. Addie .2005 .Experimental study on erosive wear of some metallic materials using coriolis wear testing approach. Journal of wear.258: 458-469.

[11] Tian Harry H., Graeme R. Addie . 2005. Experimental study on erosive wear of some metallic materials using corioliswear testing approach. Journal of wear .258 : 458-469.

[12] Tian Harry H., Addie Graeme R., Pagalthivarthi Krishnan V. 2005. *Determination of wear coefficients for erosive wear prediction through coriolis weartesting*.Journal of wear. 259:160-170.

[13] Chen, R.C. and Kadambi, J.R. 1994. Experimental and Numerical Studies of Liquid-Solid Pipe Flow.ASME-FED.189: 123-135.l

[14] PagalthivarthiK.V.,Desai P.V. and Addie G.R.1990.*Particulate motion and Concentration field in Centrifugal pump.* Particulate Science and Technology.8:77-96

[15] Brown, G. J. 2002.*Erosion prediction in slurry pipeline tee-junctions.*Applied Mathematical Modelling. 26**:**155-170.