Available online at http://www.journalcra.com

ISSN: 0975-833X

International Journal of Current Research
Vol. 10, Issue, 11, pp.75069-75076, November, 2018

DOI: https://doi.org/10.24941/ijcr.32972.11.2018

INTERNATIONAL JOURNAL OF CURRENT RESEARCH

RESEARCH ARTICLE

PREDICTION OF EROSION RATE IN CONE FLOW METER USING COMPUTATIONAL FUID DYNAMICS (CFD)

*Ranjith Khumar Shanmugasundaram

Kumara Guru College of Technology, Coimbatore

ARTICLE INFO

Article History:
Received 19th August, 2018
Received in revised form
26th September, 2018
Accepted 04th October, 2018
Published online 29th November, 2018

Key Words:
Computational Fluid Dynamics, Erosion rate, Cone flowmeter, Discrete Phase Modeling.

ABSTRACT

Erosion is one of the major problems in oil and gas industries. For example, on transporting oil, gas or water in the presence of sand particles through a long pipeline at different fluid properties may lead to material loss and decrease in the flowrate. Erosion occurs in different types on basis of the concentration and size of the particle and flow parameters. So it is very essential to predict the erosion rate in pipelines in order to have the better flow rate at different operating conditions. Computational Fluid Dynamics is one of the most commonly used tool for predicting the erosion rate. The aim of the project work is to study the erosion rate at varying conditions. First, the model of the cone flowmeter was designed using PTC Creo 3.0. And then the surface mesh of the cone flowmeter was generated using ANSA 15.1.1 and the volume mesh was generated in Turbo Grid. And using Dense Discrete Phase Model (DDPM) the results were generated in ANSYS FLUENT 16.0

***Citation: Ranjith Khumar Shanmugasundaram.* 2018.** "Prediction of erosion rate in cone flow meter using computational fuid dynamics (cfd)", *International Journal of Current Research*, 10, (11), 75069-75076.

INTRODUCTION

Cone flowmeter is a device which measures flow over wide range of Reynolds number and at different fluid conditions. It is mainly use in petroleum industries, oil and gas industries. These Cone flowmeters are subjected to erosion. Erosion can be defined as material loss resulting from impact of solid particles on the material surface. The consequences of the erosion can be crucial. This erosion may cause deflections in the flow measurement. So it is necessary to predict the erosion rate accurately. The most common way of predicting the erosion rate is particle impact based method, which is done by calculating material removal for the impact velocity at an angle. But method is risk in oil and gas industries since the particles are suspended in the fluid and not accurate as well. Computational Fluid Dynamics is now the best way to predict the erosion rate accurately and quickly. It is less hazardous and can be done for the varied conditions. Before thedesign of cone flowmeter a brief introduction was given to Computational Fluid Dynamics and Erosion.

Theory of erosion: As mentioned earlier erosion is the material loss due to the impact of the solid particles. Erosion rate is defined the amount of material removed per unit area. The erosion rate depends on the following parameters.

- Angle of impingement
- Impact velocity
- Particle diameter
- Particle mass
- Collision frequency between particles and solid walls
- Coefficients of restitution for particle-wall collision
- The erosion rate can be calculated by the given equation.

$$E = \frac{m_p k v_p^n F(\alpha)}{\rho_t A_t}$$

***Corresponding author:* Ranjith Khumar, S.,**
Kumara Guru College of Technology, Coimbatore.

- m_P= mass of the particle (kg/s)
- K,n = material constant ((m/s)^-n)
- Vp= velocity of the particle (m/s)
- ρ t= density of target particle (kg/m3)
- At = Surface area of target (m2)

$$F(\alpha) = \sum (-1)^{i+1} A\, i^{\left(\frac{\alpha\pi}{180}\right)^{i}}$$

- Ai = Area of target (m2)
- A = Angle of attack
- $F(\alpha)$= Implied angle function

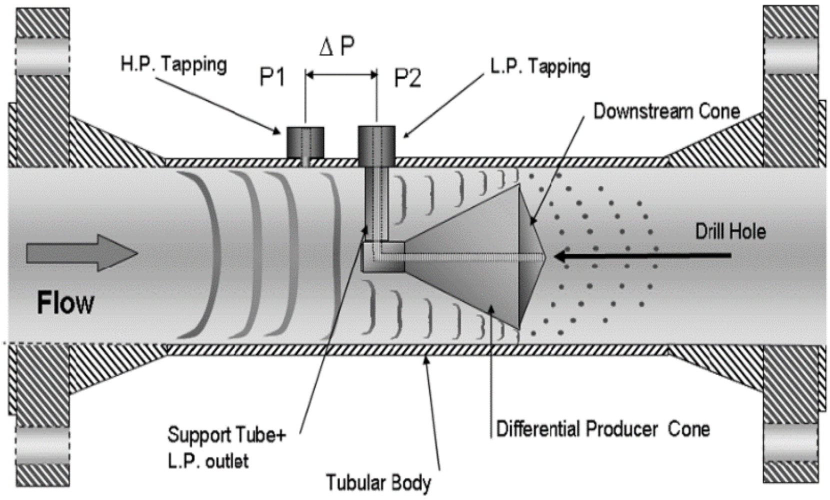

Fig 1. Differential Pressure Cone Flowmeter

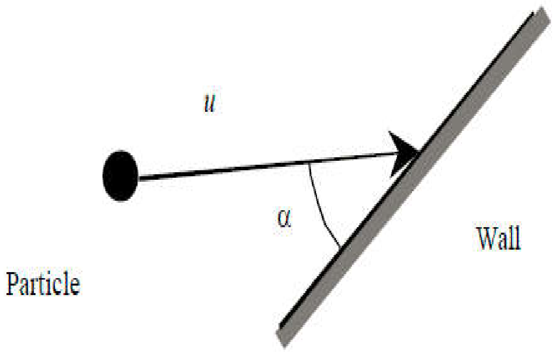

Fig. 2. Impact of the particle on flowmeter

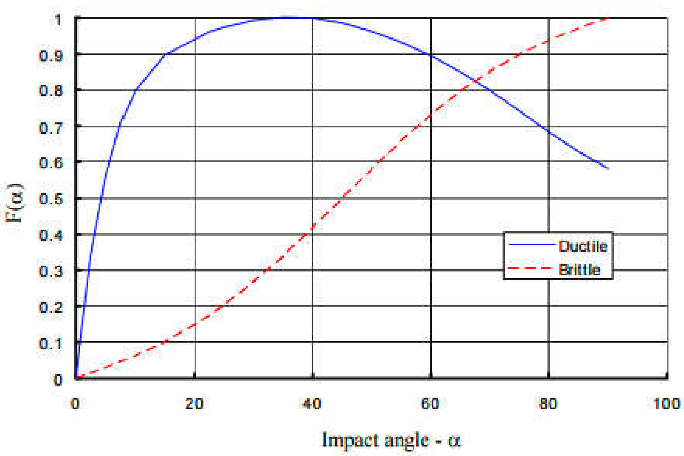

Fig. 3. The impact angle variation for Ductil and Brittle materials

Theory of computational fluid dynamics: Computational Fluid Dynamics is a set of numerical methods that are applied to obtain approximate solutions of problems on fluid dynamics and heat transfer. Fluid dynamics involves the interaction of fluid particles with other solid or fluid particles. The concept of fluid mechanics is in practice since the Buoyancy Principle given by Greek Mathematician Archimedes (285-212BC). This CFD technique is very powerful and it is being applied both in Industrial and Non Industrial applications. Some of the application of CFD includes aerodynamics, hydrodynamics, power plant, turbo machinery, marine engineering, biomedical engineering, power plants and lots more. A fluid flow generally depends upon velocity, pressure, temperature and density. It is necessary to predict these properties in order to predict the fluid flow. After a long research Navier and Stokes had developed equations involving these properties based on the conservation laws, these equations are known as Governing equations or Naviers-Stokes equations. Conservation of mass

According to the conservation of mass the rate of increase of mass in fluid element is equal to the net rate of flow of mass into the fluid element. The equation obtained by this principle is,

$$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}+\frac{\partial \rho}{\partial t}=0$$

Conservation of momentum: According to the conservation of momentum rate of increase of the momentum of the fluid particle is equal to the sum of the forces on the fluid particle. The equations obtained by this principle is,

X momentum

$$\frac{-\partial \rho}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)=\rho\left[\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right]$$

Y momentum

$$\frac{-\partial \rho}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)=\rho\left[\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right]$$

Z momentum

$$\frac{-\partial \rho}{\partial w}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)=\rho\left[\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right]$$

Where ρ is density, u,v and w are velocities in x, y and z respectively. Main aim of CFD technique is to solve these parameters using mass and momemtum equations.

Conservation of energy: According to the conservation of energy the rate of increase of energy of the fluid particle is equal to the sum of net rate of heat added to the particle and the net rate of work done on the fluid particle.

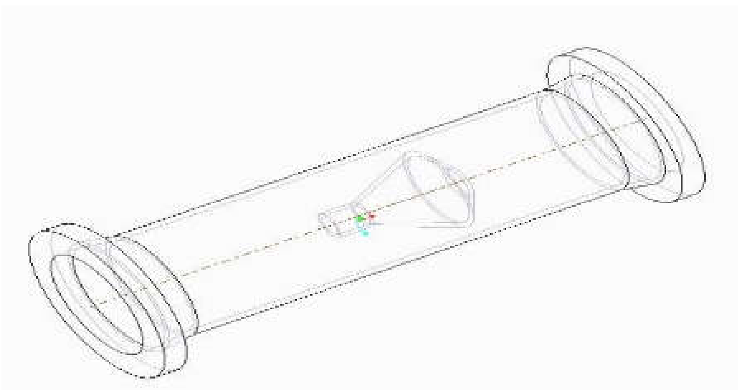

Fig. 4. Creo model of Cone flowmeter

Design of cone flowmeter: The design of flow meter was carried out in Creo3.0 software. The dimensions was given in terms of inches. The diameter of the pipe in 6 inches and the length of the pipe in 24 inches. The diameter of the cone flow meter at the inlet is 2 inches and the impact angle is 154 degree. The total length of the cone flowmeter is 6 inches. The design of the cone flowmeter using creo part diagram is shown below.

Surface mesh of cone flow meter: Thesurface mesh was created using ANSA 15.1. The PID was given for the inlet and the outlet region. The triangular and quad was generated all over the domain. There were 1500 quad at the surface and 223932 triangular meshes generated inside the domain. As a total 225432 surface mesh has been generated.

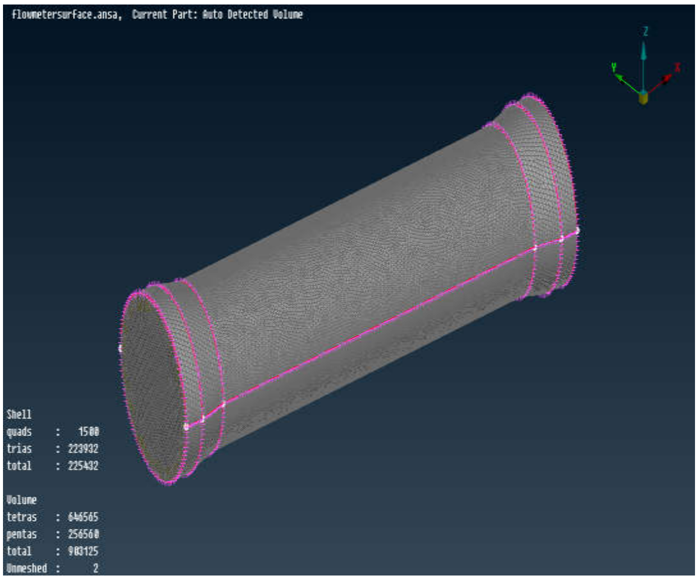

Fig. 5. Ansa Surface mesh of Cone flowmeter

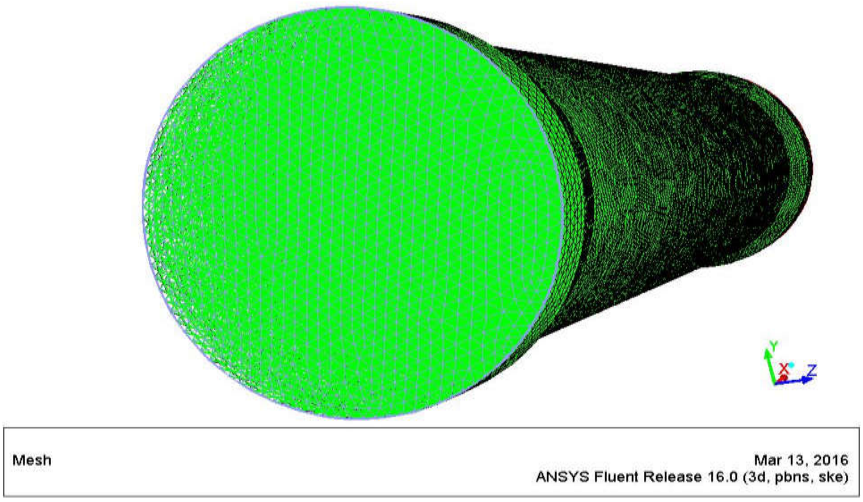

Fig 6. Ansys fluent volume mesh of cone flowmeter

Volume mesh of cone flow meter: The volume mesh was generated using ANSYS Turbo Grid (T Grid). The prisms were made to form at the surface and the tetrahedrons were formed inside the domain. In order to use K € model the y + is made to form within the viscous sub layer. The Volume mesh of our geometry is shown below. As a total 903125 volume meshes were formed

Operating conditions

- FluidFluid Water
  Fluid viscosity 1.003e-3 [kg/ms]
  Fluid density 998.2 [kg/m3]
- Solid particleSolid Sand
  Density 2600 [kg/m3]
  Diameter 1[mm]
  Volume fraction particles 0.1338262 [-]
  Sand particle definition Coarse
- Multiphase model Multiphase model Eulerian
- Eulerian parameter DDPM
- Turbulencemodel Turbulencemodel Realizable k-ε
- Wall functions Standard
- Discrete Phase Model
  Discrete phase model On
  Update DPM source Every 10 iteration
  Virtual mass force Yes
  Pressure gradient force Yes
  Drag force Morsi and Alexander

Scale flow rate by face area Yes

- Boundary conditions
  Inlet mass flow 1 [kg/s]
  Outlet relative pressure 0 [Pa]
  Wall Boundary No slip, smooth walls
  Erosion model DNV
  Solution method scheme Phase coupled SIMPLE

RESULTS

The above diagram shows the contours of the DPM erosion rate in terms kg/m2-s. Since the erosion depends on mass of the particle, density of the particle and target material, mass flow rate and impact angle, it is seen that the area at which the angle deviates more has the more erosion rate compared to other area. The flowmeter pipe has more erosion due to high velocity impact on the surface compared to other.

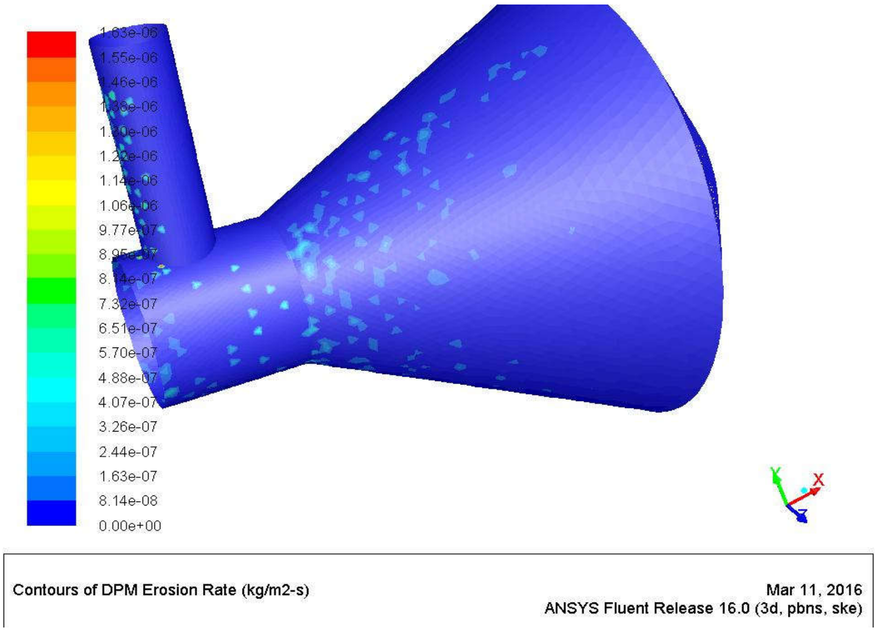

Fig. 7. Contours of DPM erosion rate (filled)

Contours of dpm erostion rate (unfilled)

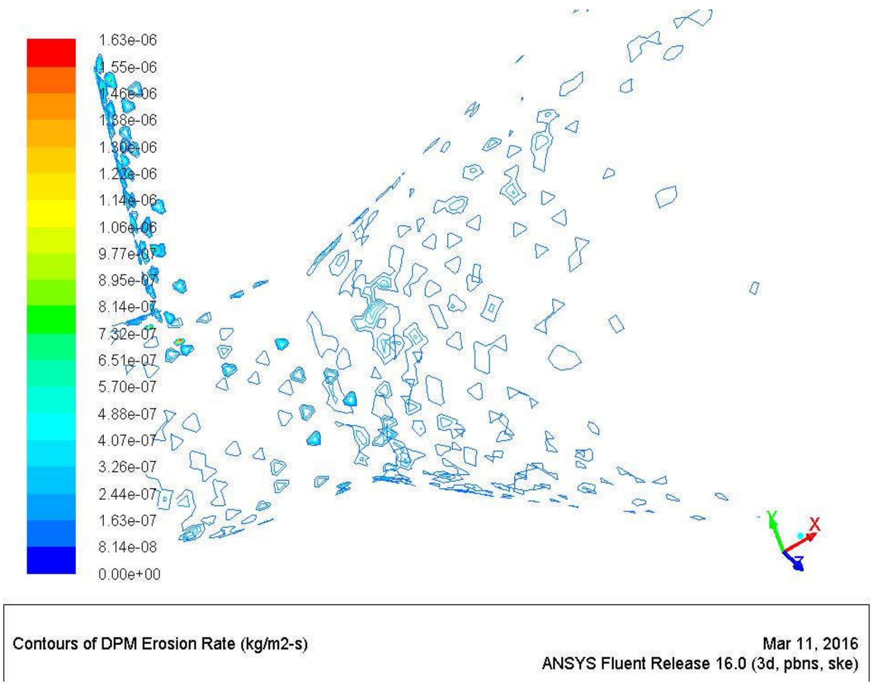

Fig 8 Contours of Erosion rate (unfilled)

Variation of erosion rate by position: The graph shows the variation of erosion rate for each positions of the domain. It is seen that that the presence of flowmeter increases the erosion rate at that position due to the impact of solid particles.

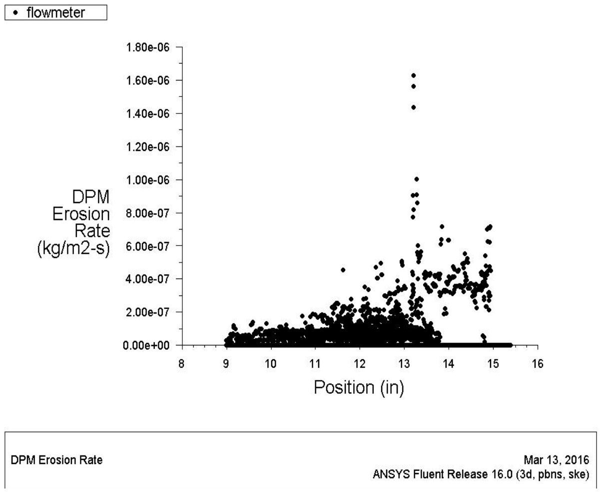

Fig. 9. Position Vs DPM Erosion Rate graph.

Contours of dpm accreation rate: The above diagram show the accretion rate in the flowmeter. This describes the amount of the slurry particles (i.e.) Sand particles that is accumulated on the surface of the cone flowmeter.

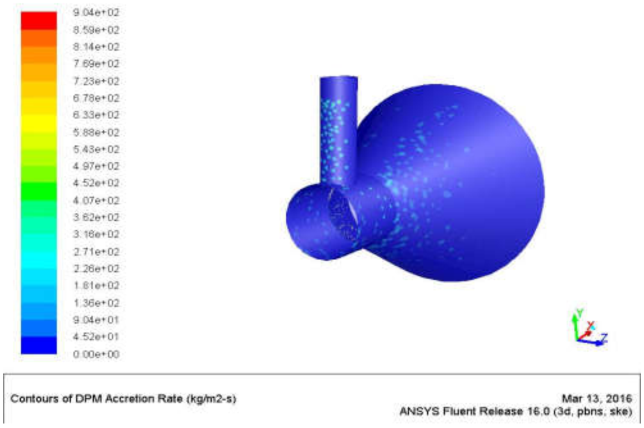

Fig. 10. Contours of DPM Accretion rate

Contours of velocity magnitude: The above diagram shows the contours of velocity magnitude. The velocity is maximum inside the flowmeter, this is due to decrease in the area which obeys the continuity equation. The velocity also increases in the narrow area between flowmeter and the outer wall.

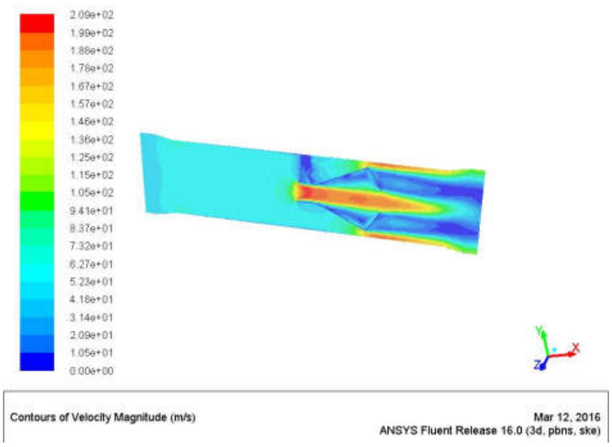

Fig. 11. Contours of Velocity magnitude

Vectors of velocity magnitude: The above diagram shows the vectors of the velocity. This shows the path of water flow inside the cylinder. At the wall the velocity becomes zero due to the formation of boundary layer. At the flowmeter due to the stagnation, the velocity becomes zero.

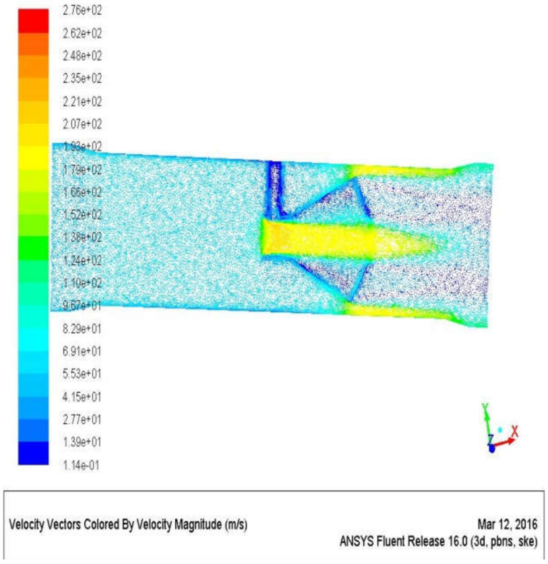

Fig. 12. Vectors of velocity magnitude

Contours of static pressure: Static pressure is the pressure exerted by a fluid that is not moving or flowing. Due to high velocity after entering the flowmeter the effects of dynamic force is less and hence the static pressure is more before the fluid entering into the flowmeter.

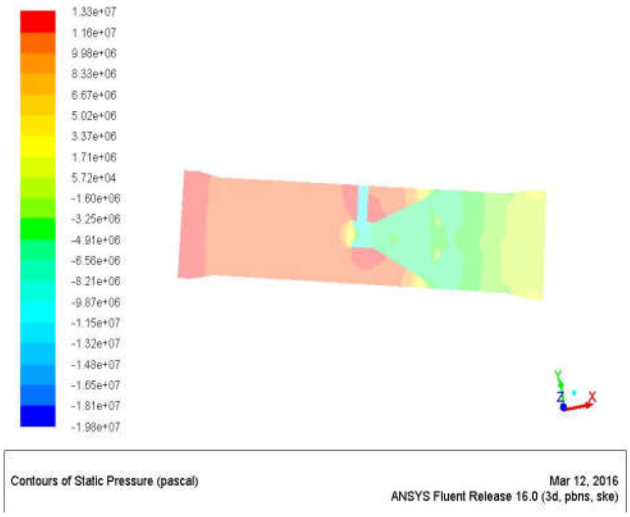

Fig. 13. Contours of Static Pressure

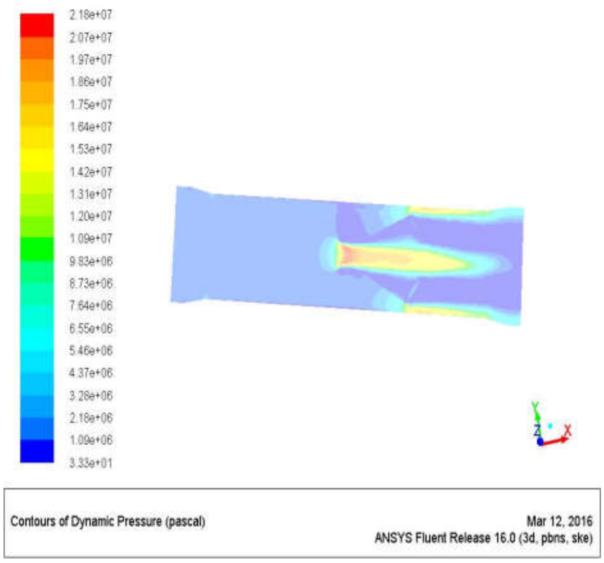

Fig. 14. Contours of Dynamic Pressure

Contours of dynamic pressure: Dynamic pressure is the kinetic energy per unit volume of a fluid particle. Since the mass is constant and it depends on velocity, as said in velocity profile due to the contraction of are the velocity increases, and so the dynamic pressure gets increased where the velocity is increasing.

Conclusion

The Erosion rate of cone flowmeter was thus predicted. The analysis can be done with different erosion model and the results can be validated. Based on the results the part of the object which is subjected to erosion can be improved. By varying different fluids and fluid conditions the erosion rate can be checked. Steps can be taken to improve results and generate coding. This could also be done with other softwares and can be validated with the results.

REFERENCES

ANSYS Fluent Support 2015.

Bredberg, J. 2000. 'On the Wall Boundary Condition for Turbulence Models', *Chalmers Internal Report 00/4.* pp. 11-12. Available at:http://www.tfd.chalmers.se/~lada/postscript_files/jonas_report_WF.pdf

Dosanjh, S. and Humphrey, J. A. C. 1985. 'The influence of turbulence on erosion by a particle-laden fluid jet', *Wear, vol. 102* (no. 4) pp. 309-330. Available at:http://www.sciencedirect.com/science/article/pii/0043164885901759 (20.03.2015)

Kosel, T.H. 1992. 'Solid Particle Erosion', *ASM Handbook, Vol. 18,* pp. 199-213. Available at: http://app.knovel.com/web/toc.v/cid:kpASMHVFL2/viewerType:toc/root_slug:asmhandbook- volume-18/url_ slug: kt00 7PXRS6

Mali, T., Khudabadi, V., Rana, A.S., Vijay, A. and Adarsh, M.R. 2014. *Slurry-Flow Pressure Drop in Pipes with Modified Wasp Method.* SME Annual Meeting/Exhibit. Salt Lake City.

Versteeg, H.K. and Malalasekera, W. 1995. *An introduction to Fluid Dynamics: The Finite Volume Method.* England: Longman Scientific and Technical.

Wasp, E. J. 1977. *Solid-liquid flow - Slurry Pipeline Transportation*. Series on Bulk Material handling Vol. 1. Germany: Trans Tech Publications.

White, F.M. 1999. *Fluid Mechanics*. Fourth Edition. Singapore: McGraw-Hill Book Co. Anderson. Computational fluid dynamics: the basics with applications. Tata McDraw-hill edition.

Det Norske Veritas. 2007. RP O501 – Revision 4.2. *Erosive Wear in Piping Systems.* Høvik: Det Norske Veritas.

ANSYS Help Viewer 2013. *ANSYS Help Viewer,* Verison 15.0.0.

****