TWO-PHASE FLOW CFD MODELING FOR EVALUATION OF PARTICULATE EROSION

Stefan NICOLICI[1], Ilie PRISECARU[2], Daniel DUPLEAC[3]

În această lucrare codul de calcul Ansys CFX a fost utilizat pentru simularea curgerii bifazice în coturile de conductă supuse riscului de eroziune cu particule lichide sau solide. Interacţiunea dintre fluidul purtător (aer sau vapori) şi particule este modelată considerând cuplarea bi şi unidirecţională şi efectele turbulenţei fazei continue asupra particulelor prin forţa de dispersie. Obiectivul principal a fost de a obţine caracteristicile de impact ale particulelor asupra peretelui exterior al cotului: viteză, unghi şi frecvenţă de impact. Transferul de impuls dintre particule şi fluidul purtător este calculat folosind două metode de cuplaj. Influenţa anumitor parametrii caracteristici (dimensiunea particulelor, fluxul masic de particule, viscozitatea fazei continue) a fost cercetată.

In this paper the CFX Ansys Code was employed for simulation of two-phase flow in pipe bends subjected to erosion due to liquid or solid particles. The interaction between carrier fluid (air or steam) and particles is modeled considering full and one-way coupling and the turbulence effect of the continuous medium on particles through the particle dispersion force. The main objective was to obtain the impact characteristics of the particles upon the bend outer wall: velocity, angle and frequency of impact. The interactional force between carrier and particles is taken into account by momentum transfer in an Eulerian-Lagrangian approach. The influence of some phenomena parameters (particle dimension, particle density, carrier fluid viscosity) was investigated.

Keywords: Two-phase flow, Lagrangian-Eulerian method, particulate erosion

1. Introduction

Increasingly severe requirements for materials in high performance applications have reemphasized the need for fundamental understanding of the erosion process associated with particle impingement [1]. Experimental results and theoretical concepts are providing a framework for development of useful methods of analysis, characterization, and prediction of the response of materials

[1] PhD. Student, Nuclear Engineering Department, Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania, e-mail:nicolici.stefan@gmail.com
[2] Prof., Nuclear Engineering Department, Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania, e-mail:prisec@gmail.com
[3] Prof., Nuclear Engineering Department, Power Engineering Faculty, University POLITEHNICA of Bucharest, Romania, e-mail: danieldu@cne.pub.ro

under solid and liquid particle impact for a wide range of hostile environments. However, the interactions among the most parameters involved are very complex.

Based on the latest domestic and international knowledge, causes of the wall thinning are investigated and analyzed into mainly two causes as follows [2]; one is “FAC (Flow Accelerated Corrosion)” - the electro-chemical action with liquid flow. Other is “Erosion” - the mechanical action on the wall by the power of liquid droplets, solid particle or cavitations in the system.

In the power industry, metallic surface erosion by particle impact can substantially shorten the lives of pipelines, heat exchanger systems, and turbo machinery surfaces [3]. Investigations into the field of erosion-corrosion are typically daunted by the huge amount of experimental parameters which may have an effect on this synergistic damage mechanism, including: flow conditions, composition of the structural material, chemistry of the flowing system and temperature. The wall thinning phenomenon has been often found at the outer wall of bent pipes in the power engineering industry. The phenomena have been considered to be primarily due to corrosion and erosion mechanisms.

Particle impact erosion is a phenomenon which occurs due to solid and liquid particles included in high-speed two-phase flow when it comes to the area where the direction of the flow changes [4]. The factor of the phenomenon is a difference of mass. The mass of the particle is larger than that of the steam/gas, therefore continuous phase can flow along the pipe wall but the dispersed phase impinges against the wall. The impact power in the collision once is little, but the impact power accumulates and material damage occurs.

From the point of flow pattern view, erosion rate is dependent on a number of factors including particle size, impact velocity, impact frequency, particle material and gas density and viscosity [5, 6]. As many of these values are unknown for field situations, it is very difficult to predict the rate of erosion. It should also be noted that control of many of these factors in laboratory-based test is problematical. Therefore great care is required when extrapolating lab test results to field conditions [7]. In this context, Computational Fluid Dynamics (CFD) can be used as a powerful tool to investigate the particle erosion material degradation.

For the numerical simulation, in the past, turbulence modification due to presence of particles was made using an Eulerian approach [8]. On the other hand, most of recent investigations follow the Lagrangian approach. Particle transport modeling is a type of multiphase model, where particulates are tracked through the flow in a Lagrangian way, rather than being modeled as an extra Eulerian phase [9]. The full particulate phase is modeled by just a sample of individual particles. The tracking is carried out by forming a set of ordinary differential equations in time for each particle, consisting of equations for position, velocity, temperature, and masses of species. These equations are then integrated using a simple integration method to calculate the behavior of the particles as they traverse the flow domain.

2. Computational models and methods

2.1. Computational domain

As discussed above, material thinning phenomena occurs in those elements where the flow presents a direction change. In this paper, the pipe element chosen to study the two phase flow pattern is a pipe bent. The computational domain and mesh are shown in Figure 1.

The research field of computational fluid dynamics is based on the basic conservation equations for mass, momentum, energy (and species concentration). By solving a set of nonlinear partial differential equations, more accurate results can be obtained, e.g. transient behavior, movement due to convection and transport due to laminar and turbulent diffusion.

Numerical accuracy and resolution are important issues in CFD calculations. Grids consisting of about 460 000, 239 000, 133 000 and 72 000 cells were tested for the same case. The average pressure drop from inlet to outlet is observed to change less than 1% among the four grid density solutions, and finally we adopted 133 000 cells as our computational mesh.

2.2. Governing Equations

The aim of computational fluid dynamics is to numerically solve the partial differential equations that govern fluid flows. In all two-phase cases the flow consists of a continuous phase, which may be gaseous or liquid, and one or more dispersed phases in the form of solid particles, liquid droplets or gas bubbles. In general, the motion of the dispersed phase will be influenced by that of the continuous phase and vice versa via displacement and inter-phase momentum, mass effects. In our simulations, the governing equations for the carrier phase are expressed in the Eulerian form and are suitably modified to take into account the presence of the dispersed phase. The interactions between the fluid and droplets are calculated using two different coupled methods as described in the following.

Two general equations are available for the continuous phase simulations without heat transfer, namely [10]:

- The mass of a fluid is conserved (continuity):

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \vec{v}) = 0 \tag{1}$$

- The rate of change of momentum equals the sum of the forces on a fluid particle (Navier-Stokes, Newton's 2nd law):

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla(\rho\vec{v}\vec{v}) = -\nabla p + \nabla(\bar{\bar{\tau}}) + \rho\vec{g} \quad (2)$$

in which τ represent the stress tensor given by:

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

Within the particle transport model, the total flow of the particle phase is modeled by tracking a small number of particles through the continuum fluid. The particles could be solid particles, drops or bubbles. The application of Lagrangian tracking in CFX involves the integration of particle paths through the discretized domain [11]. Individual particles are tracked from their injection point until they escape the domain or some integration limit criterion is met. Each particle is injected, in turn, to obtain an average of all particle tracks and to generate source terms to the fluid mass, momentum and energy equations. The particle momentum equation is:

$$m_p \frac{dv_p}{dt} = F_{all} \quad (4)$$

with F_{all} being the sum of all forces acting on a particle given in Table1.

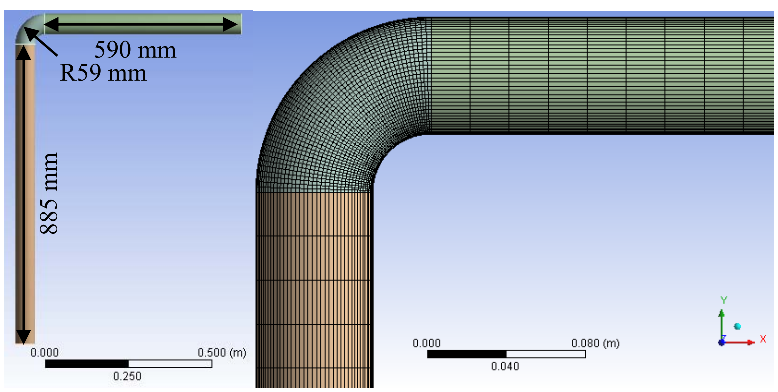


Fig. 1. Schematic diagram and bent section mesh

The flow prediction of the two phases in one-way coupled systems is relatively straightforward. The fluid flow field may be calculated irrespective of the particle trajectories. One-way coupling may be an acceptable approximation in flows with low dispersed phase loadings where particles have a negligible influence on the fluid flow.

Two-way coupling requires that the particle source terms are included in the fluid momentum equations. The momentum sources could be due to turbulent dispersion forces or drag. The particle sources to the fluid momentum equations are obtained by solving transport equations for the sources. The generic equation for particle sources is:

$$\frac{dS_p}{dt} = C_S \varphi_p + R_s \tag{5}$$

where $C_S \varphi_p$ are the contributions from the particles that are linear in the solution variable and R_s contains all other contributions.

Table 1

Forces acting on a particle

Force	Expression
Drag	$\vec{F}_D \propto C_D A_F \left\|\vec{U}_p - \vec{U}_F\right\| \left(\vec{U}_p - \vec{U}_F\right)$
Buoyancy	$\vec{F}_B \propto \left(\rho_p - \rho_C\right)\vec{g}$
System rotation	$\vec{F}_R \propto 2\vec{\Omega} \times \vec{U}_P \rho_p - \vec{\Omega} \times \vec{\Omega} \times \vec{r}_p \rho_p$
Added mass	$\vec{F}_V \propto \frac{1}{2} M_F \left(d\vec{U}_F / dt - d\vec{U}_p / dt\right)$
Pressure gradient	$\vec{F}_P \propto M_F d\vec{U}_F / dt = M_F \rho_F \nabla p$

2.3. Boundary Conditions

In our simulations we assume the carrier fluid is air. The governing equations are elliptic in the Cartesian coordinates; hence the boundary conditions are required for all boundaries of the computational domain (Figure 2). The required conditions are described for the three different parts, which are inlet, wall and outlet, respectively.

The implementation of pipe wall boundary is treated as the normal no-slip wall and the wall surface is supposed to be smooth. The air density is 1.185kg/m^3 at room pressure (1.01×10^{5}Pa) and room temperature (298K). A uniform velocity profile was used at the inlet, the mass flow rate being 0.0323587 kg/s corresponding to a velocity of v = 10 m/s. The outlet is prescribed using a relative average pressure of 1 bar.

The mass flow rate of the particles is 0.001618 kg/s with the velocity of 10m/s at the entry. The particle density is 997kg/m^3, which is around 841 times larger than carrier (air), this large density ratio will bring strong effects to the particles trajectories. The impact of particles, droplets or bubbles on rigid surfaces may produce a wide variety of consequences, according to the size, velocity and material of the impacting elements and the nature of the surface. For example, droplets may adhere, bounce or shatter, and the liquid deposited on the surface may retain its droplet form or merge into a liquid film. In our numerical simulations, the treatment of wall and droplet interaction is as the perfect rebound function.

3. Results

3.1. Carrier Fluid Streamlines and Droplet Trajectories

In uncoupled flows the feedback from the dispersed to the continuous phase is negligibly small, as the one-way method explanation indicates. By contrast, coupling refers to the situation where the continuous phase significantly influences the motion of the dispersed phase and vice versa.

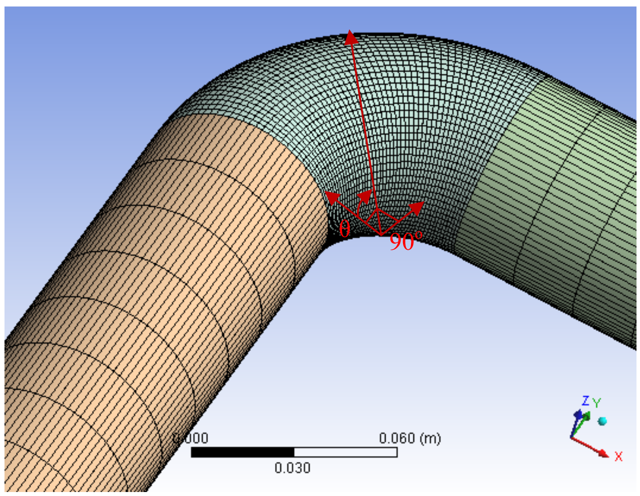


Fig. 2. Computational Coordinate and the θ Definition

3.1.1. One-way Calculation

In the one-way calculations, the droplets trajectories of different droplet sizes are shown in Fig. 3 (a, b, c). Different droplet sizes of 10µm, 50µm, 100µm are explored to discuss the influence of droplet size. Based on the same inlet flow conditions, the effect of droplet size on the trajectory can be observed. There is no change in carrier fluid streamlines (Fig. 3d) for different droplet sizes because there is no contribution to the source term for the continuous phase equation.

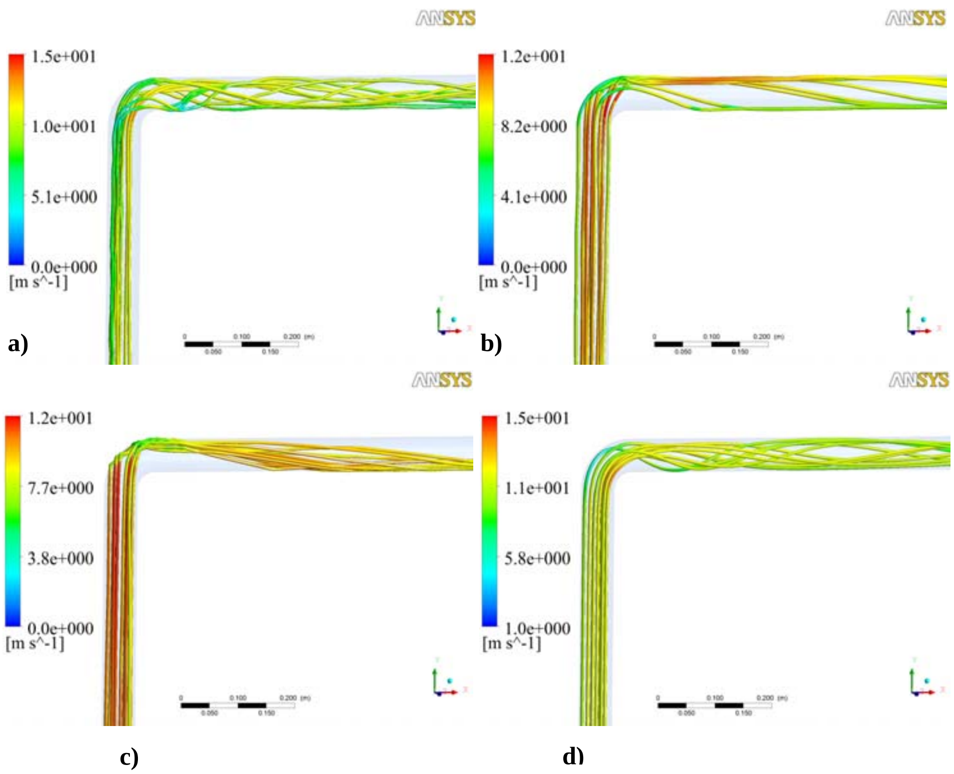


Fig. 3. Droplet Trajectories (a - 10μm, b - 50μm, c - 100μm) and carrier fluid streamlines (d) (One-way Calculation)

3.1.2. Two-way Calculation

Figure 4 shows the outlet fluid velocity on y position in a two-way calculation. Compared with the one-way calculation, the droplet average momentum is performed as a source term in the momentum equation of the carrier phase in two-way calculation, the carrier streamlines differ due to distinct droplet sizes (10μm, 50μm and 100μm), resulting in different velocity profiles which are shown in Fig. 4. Both the carrier and particles are influenced by each other, so the droplet trajectories are similar nevertheless different from those in one-way calculations and the carrier fluid is somewhat influenced by the droplets due to the two-way coupling fluid-droplet system.

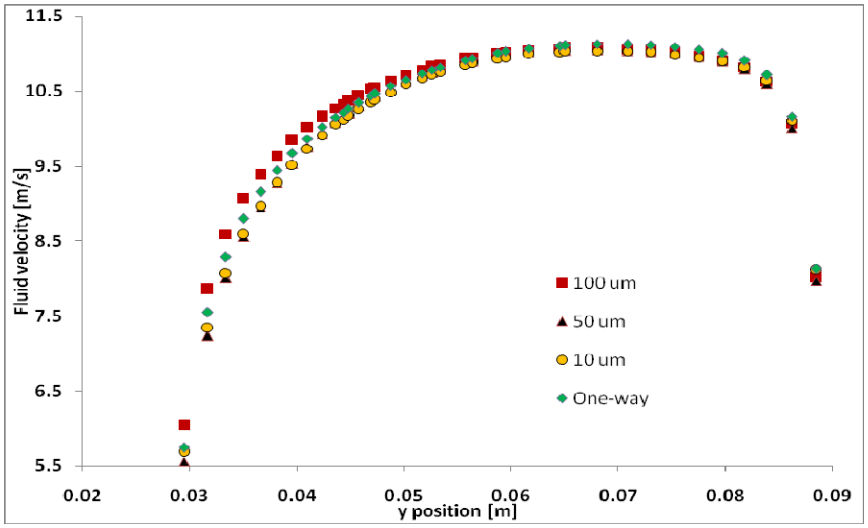
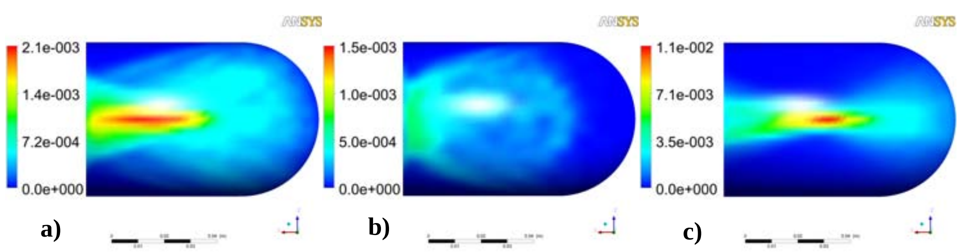


Fig. 4. Fluid velocity at outlet on y direction for different particle sizes and one-way simulation

3.2. Buoyancy influence on particle trajectories

The effect of gravity upon the particle trajectories can play an important role on material maximum erosion rate. The buoyancy of the dispersed phase is considered through buoyancy force given in Table 1, being proportional to density difference between carrier and particles densities. Several cases were studied for the same inlet conditions and materials, but imposing different directions of the gravitational acceleration. The results are presented in Fig. 5. We can observe that not only the position of the maximum particle volume fraction is changed, but also its magnitude.



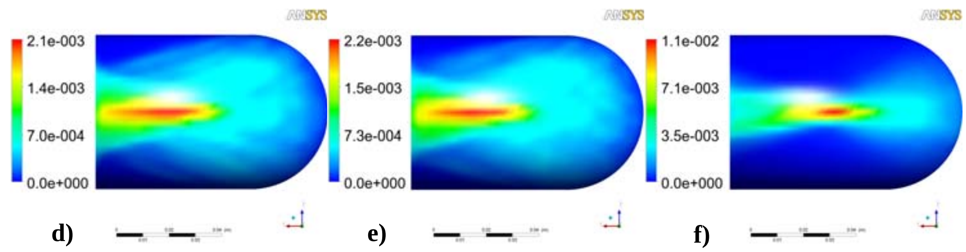


Fig. 5. Particle volume fraction on outer wall for different gravity directions: a) no gravity, b) +x direction, c) -x direction, d) +y direction, e) -y direction, f) +z direction

3.3. Materials properties

Both particle material and carrier fluid properties affect the particle trajectories. For example, denser particles imply higher particle momentum and a carrier fluid drag with a smaller contribution. Four particle densities were used: 1000 kg/m^3, 3000 kg/m^3, 5000 kg/m^3 and 7000 kg/m^3. The particle trajectories are presented in Fig. 5 (a, b, c, d). Also, the viscosity of the continuous phase affects the drag force that is most conveniently expressed in terms of the dimensionless drag coefficient C_D [11]:

$$C_D = \frac{D}{1/2\rho_C (U_P - U_C)^2 A} \tag{6}$$

where D is the magnitude of the drag force and A is the projected area of the body in the direction of flow. In this paper Schiller Naumann drag model was employed, which gives the drag coefficient as function of Reynolds criterion calculated using carrier fluid properties and the diameter of the particle:

$$C_D = \frac{24}{\mathrm{Re}}\left(1 + 0.15\,\mathrm{Re}^{0.687}\right) \tag{7}$$

As we can notice, the carrier fluid viscosity has an influence on drag force through Reynolds number. The maximum particle volume fractions on outer bent wall for three particle diameters are presented in Figure 6.

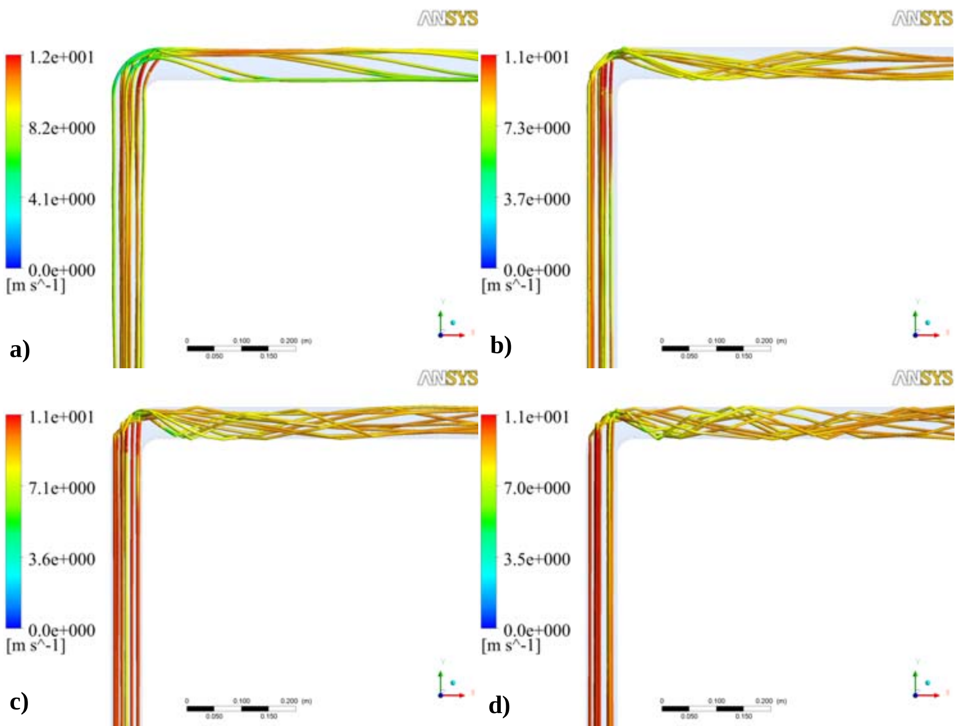


Fig. 5. Particle trajectories for different particle densities: a) 1000 kg/m^3, b) 3000 kg/m^3, c) 5000 kg/m^3, d) 7000 kg/m^3

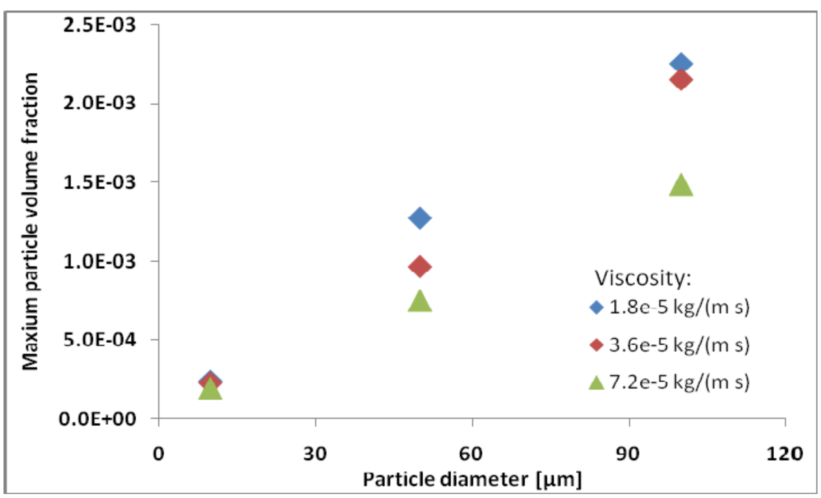


Fig. 6. Maximum particle volume fractions on outer bent wall at different carrier fluid viscosities

4. Conclusions

In this paper, three-dimensional computational fluid dynamics calculations were conducted to investigate the carrier fluid characteristics and particle trajectories in a full-scale bent pipe using one-way and two-way fluid-droplet coupled systems. The impacts of centrifugal and gravitational forces on the liquid droplet behaviors within the piping can be reasonably captured by the two-phase model. The inter-phase effects between the dispersed phase and the continuous phase are accounted for by an Eulerian-Lagrangian approach. Various parameters related to flow conditions were varied to demonstrate their influence on the erosion rates of the bent pipe. Many of these factors which control the rate of erosion, such as particle velocity or particle mass flow rate, particle diameter, impact angle and particle distribution can be studied at different flow conditions of the system, in order to reduce the wall material wastage.

The major conclusions are drawn as follows:

1. Based on the carrier streamlines and particle trajectories, the two-way calculation using the inter-phase momentum transfer system could be a more appropriate model to simulate the bent pipe wall thinning phenomena depending on particle diameter and density, and particle loading. For small particles or low particle loading the effect upon the continuous phase is negligible.
2. In order to explore the details of bent position in space, we studied the effect of different directions for gravity. The results showed that the maximum particle volume fraction and its position on outer wall changes.
3. For the materials properties evaluation, the results demonstrate that denser particles carried in lower viscosity continuous phase increase the maximum particle impact number on the bent wall.
4. A complex interaction of three forces influences the motion of particles in three-dimensional piping system: gravity, particle inertia and the drag force. Particles move to either the inner or outer walls due to a balance of particle inertia and gravity, while they are at the same time moved to the side walls due to the fluid motion.

Only one turbulence model (k – eps) was employed for the carrier phase flow, the effects of different turbulence models should be studied in the future work.

REFERENCES

[1] *Fumio Inada,* " Japanese Pipe Wall Thinning Management Based on JSME Rules and Recent R&D Studies to Enhance the Rules" Technical Meeting/Workshop on E/C Including FAC and EAC Issues in NPP, 21 - 23 April 2009, Moscow, Russia

[2] *A. Taboada, L. C. Shao,* "Piping Issues in Nuclear Systems in the USA" in International Journal of Pressure Vessels & Piping **34**, 1988, pp. 309-313

[3] *Y.M. Ferng,* " Predicting local distributions of erosion–corrosion wear sites for the piping in the nuclear power plant using CFD models" in Annals of Nuclear Energy **35**, 2008, pp. 304–313

[4] *Yuma Higashi, Tadashi Narabayashi, Yoichiro Shimazu, Masashi Tsuji Shuichi Ohmori, Michitsugu Mori, and Kenichi Tezuka*" Study on pipe wastage mechanism by Liquid Droplet Impingement Erosion " Proceedings of ICAPP '09, Tokyo, Japan, May 10-14, 2009

[5] ***Nippon Kaiji Kyokai, "Guidelines on Pipe Wall Thinning" Technical Report, September 2008

[6] *L.J.W. Grahama, D.R. Lesterb, J. Wua,* "Quantification of erosion distributions in complex geometries" in Wear **268**, 2010, pp. 1066–1071

[7] *Y. Zhan,g E.P. Reuterfors, B.S. McLaury, S.A. Shirazi, E.F. Rybicki,* "Comparison of computed and measured particle velocities and erosion in water and air flows" in Wear **263**, 2007, pp. 330–338

[8] *Christopher E. Brennen,* Fundamentals of Multiphase Flows, Cambridge University Press, 2005

[9] *Xianghui Chena, B. S. McLauryb, S. A. Shirazi,* "Numerical and experimental investigation of the relative erosion severity between plugged tees and elbows in dilute gas/solid two-phase flow" in Wear **261**, 2006, pp. 715–729

[10] *Nikolay I. Kolev,* "Multiphase Flow Dynamics", Springer Berlin Heidelberg, 2005

[11] ****Ansys INC.,* "Ansys-CFX13 User Manual", 2010