Modeling of the metal powder flow with carrier gas in coaxial nozzle for direct laser deposition process

Heng Pan and Frank Liou

Laser Aided Manufacturing Processes lab
Department of Manufacturing Engineering
University of Missouri – Rolla, Rolla, MO, 65401
Reviewed, accepted August 4, 2004

ABSTRACT

In direct laser deposition process, metal powder is directly fed with carrier gas through the coaxial nozzle into the melt pool created by the laser to form the solid parts. Many operational parameters of the process depend on the characteristic of the powder stream structure below the exit of the coaxial nozzle. In this paper, a computational approach is developed for the simulation of the gas-particle flow in the coaxial nozzle. By taking into account the nozzle geometry and operating parameters, such as width and inclination angle of powder passage and carrier gas velocity, the developed computational code allows the simulation, optimization and control of the delivery of the metal powders.

INTRODUCTION

The direct laser deposition process is an extension of the laser cladding process for rapid prototyping of fully dense metal components. It involves the supply of metallic powders into a laser-heated spot where the powder is melted and forms a melt puddle which quickly solidifies into a bead. The development of the coaxial nozzle was a major step forward in this process, since coaxial feed would make the process omni-directional and therefore easier to use [1]. Here, it is important to establish a well focused powder stream at the exit of the coaxial nozzle in order to obtain good quality and avoid waste of powders. In achieving this, a proper design of the coaxial nozzle is essential.

A schematic diagram of the coaxial nozzle used in the process is shown in Fig. 1. The carrier gas laden with metal powders is delivered through a passage formed by inner and middle nozzle body. The width and inclination angle of the passage have direct impact on the powder stream configuration. Most nozzle designs rely on experiments [1, 2, 3]. Some attempts have been made to improve the nozzle design through modeling. A mathematical model proposed by Pinkerton [4] is able to predict a powder stream profile with the main purpose of evaluating concentration distribution. This analytical model is easy to apply. However, it appears to be insufficient in assessing the effects of a more detailed geometrical variation of the nozzle on the powder stream, which is normally regarded as requiring a numerical approach. Lin described a numerical method for prediction of the powder stream structure at the nozzle exit [5]. However, a powder model in predicting dispersed metallic powder flow within the coaxial nozzle, which is accurate and efficient enough for supporting the design optimization of the nozzle, is non-existent. An attempt has been made to model particle-wall collisions with the consideration of non-sphericity [6]. However no carrier gas has been included yet. The purpose of the paper is mainly aimed to include carrier gas for completion of the simulation.

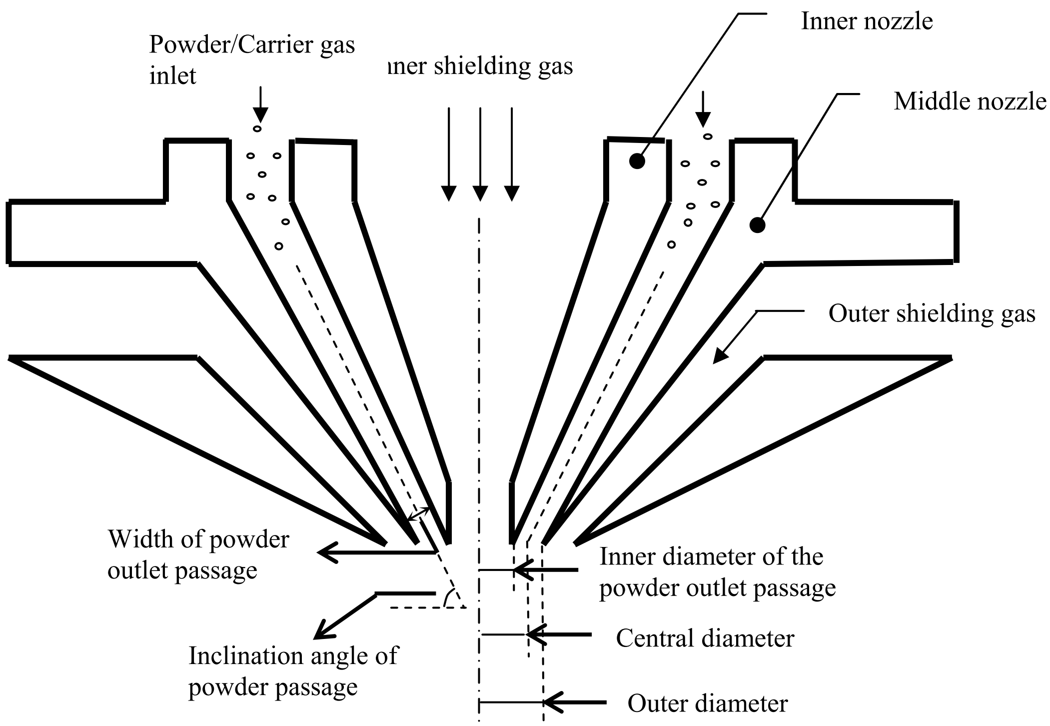


Fig. 1 Schematic of nozzle configuration.

NUMERICAL METHOD

The numerical calculations are performed by the Euler/Lagrange approach for the gas and particle phases respectively. The fluid flow calculations are based on the time averaged Navier-Stokes equations in connection with the k-ε turbulence model [7]. It is assumed that particle concentration is low enough that inter-particle interactions and modification of flow field induced by particle phase are neglected.

To better simulate the particle dispersion in the nozzle body, a specific 3-D Lagrangian particle simulation code has been developed in UMR. The code is capable of modeling particle irregular bouncing caused by non-spherical particle shape and wall roughness efficiently [6]. The carrier gas is solved numerically. Based on one-way coupling, the gas effect is superimposed on particle phase without considering particle phase effects on gas. To avoid overestimation of particle concentration near the axis, the particles must be tracked in 3-D Cartesian coordinate system. However, a full 3-D simulation takes too much computational time and will be impractical for design purposes. In order to save computational time, the following procedures are followed: the gas phase is simplified by using the axi-symmetric assumption that the variation of the flow component in the circumferential direction is negligible. The gas flow field generated by carrier within the coaxial nozzle is firstly solved using the axi-symmetric version of the Navier-Stokes equation. The flow field is then transferred to 3-D the Cartesian coordinate. The trajectory of each particle in the nozzle body is tracked in the 3-D Cartesian coordinate frame, taking into account the particle-wall collision and gas effects. Therefore, the simplified code can precisely predict particle concentration distribution at the exit of the nozzle without having to solve the full 3-D equation for both phases. It greatly reduces the computation time.

The general form of the elliptic differential equations governing an axi-symmetric turbulent and incompressible flow is given by equation (1). The source terms of the gas phase, S and effective viscosity, Γ, are summarized in Table 1 for the dependent variables, Φ.

$$\frac{\partial}{\partial z} r\left(u_z \phi - \Gamma \frac{\partial \phi}{\partial z}\right) + \frac{\partial}{\partial r} r\left(u_r \phi - \Gamma \frac{\partial \phi}{\partial r}\right) = S \tag{1}$$

The gas equations are solved using a finite volume method. Pressure-velocity coupling is realized by the SIMPLE algorithm. This method solves discrete versions of the equations by a solution of pressure-correction equation to force the velocity predicted by momentum equations to satisfy mass-flux continuity.

The motion of particles is simulated using the Lagrangian particle simulation code coupled with the flow field. The individual particle trajectories are calculated by solving the equations of motion for each particle. The resulting equations for determining the change in particle location and velocities are:

$$\frac{dx}{dt} = u_p \tag{2}$$

$$\frac{du_p}{dt} = \frac{3\rho}{4\rho_p D_p} c_{drag} (u - u_p) \,|\, u - u_p \,| + F \tag{3}$$

The drag coefficient for the particle is obtained in dependence of the particle Reynolds number Re_p,

$$c_{drag} = \frac{24}{\mathrm{Re}_p}(1.0 + \frac{1}{6}\mathrm{Re}_p{}^{1.66}) \qquad \mathrm{Re}_p \le 1000 \tag{4}$$

$$c_{drag} = 0.44 \qquad \mathrm{Re}_p \ge 1000 \tag{5}$$

$$\mathrm{Re}_p = \frac{D_p \rho \,|\, u - u_p \,|}{\mu} \tag{6}$$

where D_p is the diameter of the particle, μ is viscosity, u is gas velocity, u_p is velocity of the particle, F denotes external forces like gravity.

The particle turbulent dispersion has been modeled by sampling random Gaussian gas velocity fluctuations and accounting for the crossing trajectories and eddy lifetime effects.

The particle-wall collisions that take place in the coaxial nozzle have been modeled in a three-dimensional frame of coordinates by solving impulse equations coupled with Coulombs law of friction. Since the metallic powders used in this process are not purely spheres in morphology, a non-spherical model is applied to better simulate irregular bouncing induced by a certain degree of non-sphericity [6]. Empirical restitution, friction coefficients and the nozzle wall surface roughness model provided by Sommerfeld [8] have been used for the calculation.

The computational domain for modeling carrier gas is indicated in Fig. 2. Along the entire length of the axis, the radial velocity and the radial gradients of other variables are set to zero. Across the outlet plane, assuming that the flow there has fully developed, the axial gradients of all variables are neglected. At the entrainment boundary, axial velocity is assumed to be zero and the quantity ru_r is constant. All solid lines represent walls.

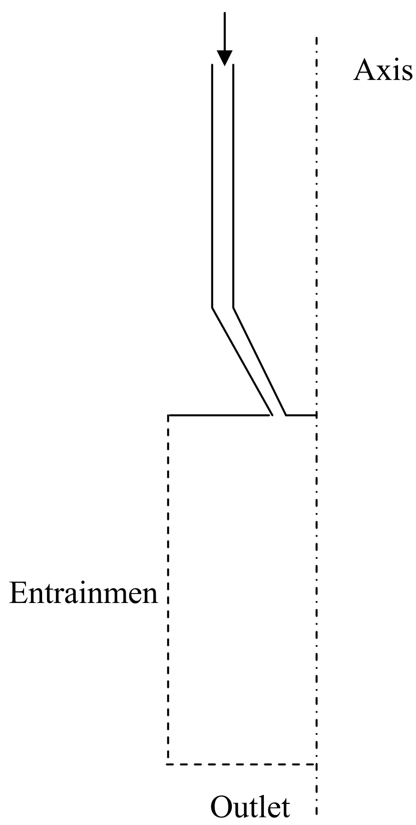


Fig. 2. Computational domain.

SIMULATION RESULTS

The model is then used to evaluate the geometry of the powder passage and carrier gas settings on the characteristics of the flow stream. Typical simulated particle concentration contour is shown in Fig. 3. The predicted changes in the powder concentration variation along the axial and radial direction under different conditions are shown in Fig. 4-6. In Fig. 4, only the inclination angle is varied. In Fig. 5, the outer diameter is changed to obtain different opening width. Fig. 6 shows that flow condition under different carrier gas velocities.

Fig. 4a shows that decreasing the inclination angle, which makes a sharper injection of powders, moves the concentration peak point towards the nozzle. The 45º nozzle exhibits an abrupt increase of concentration along the central axis, which can be called as focused stream structure, while the 60º nozzle gives rise to a column-like structure, as indicated in Fig. 3b. This structure provides a longer working distance. It should be noted that using the simple mathematical model proposed by Pinkerton's is considered to be incapable of predicting this difference. Fig. 4b indicates that the concentration variation along radial direction is similar in two nozzles. Thus it can be seen that changing inclination angle has little effect in influencing the width of the focused point.

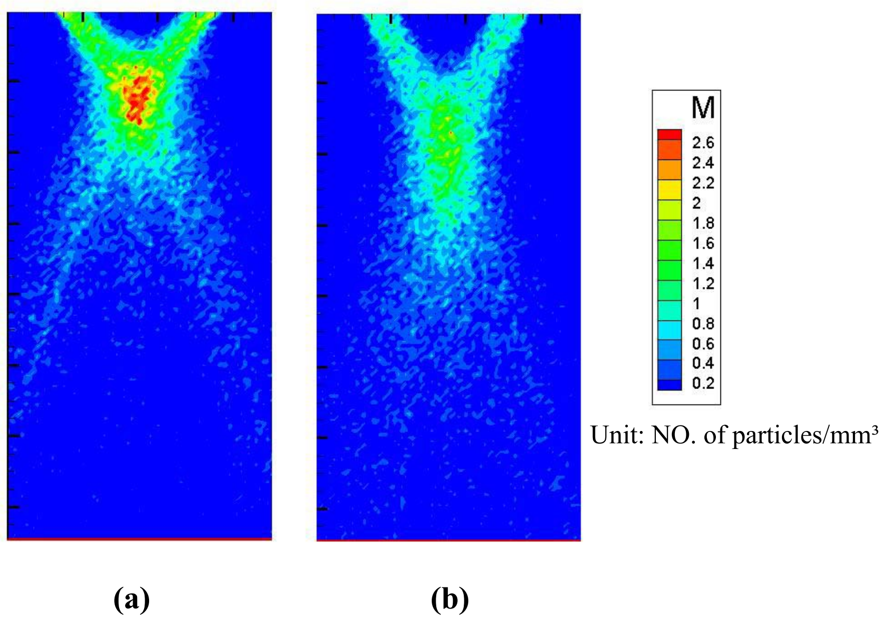
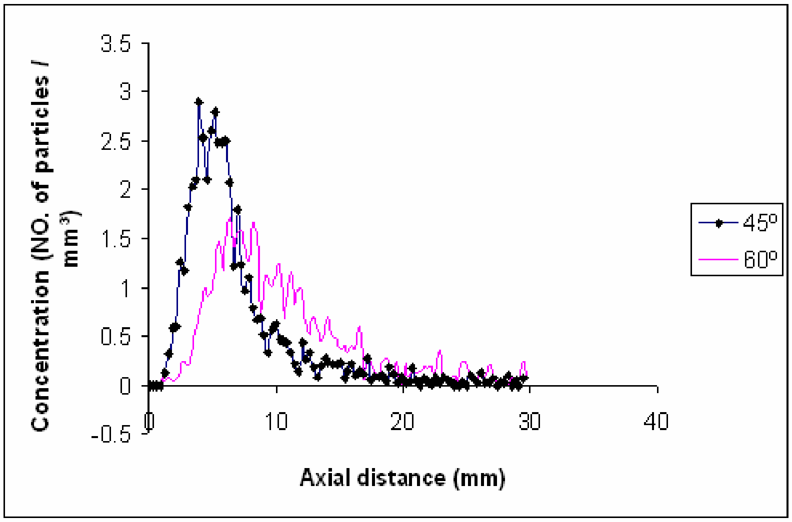
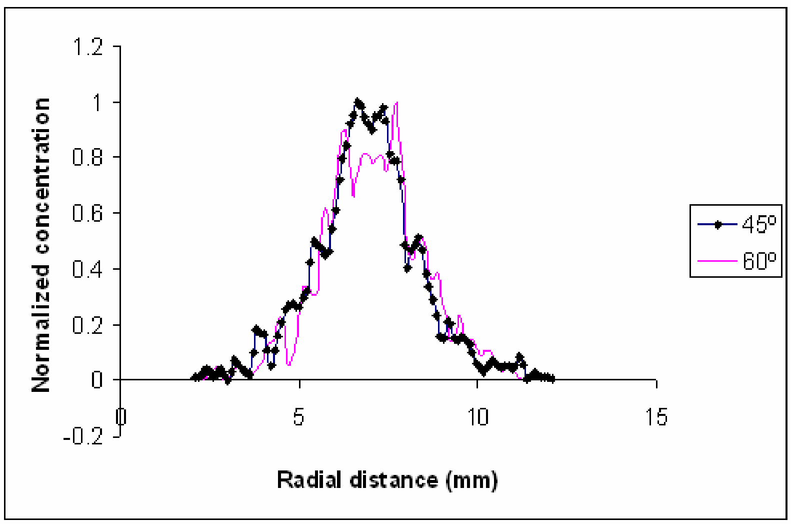


Fig. 3 Concentration contours of 45º (a) and 60º nozzles (b).



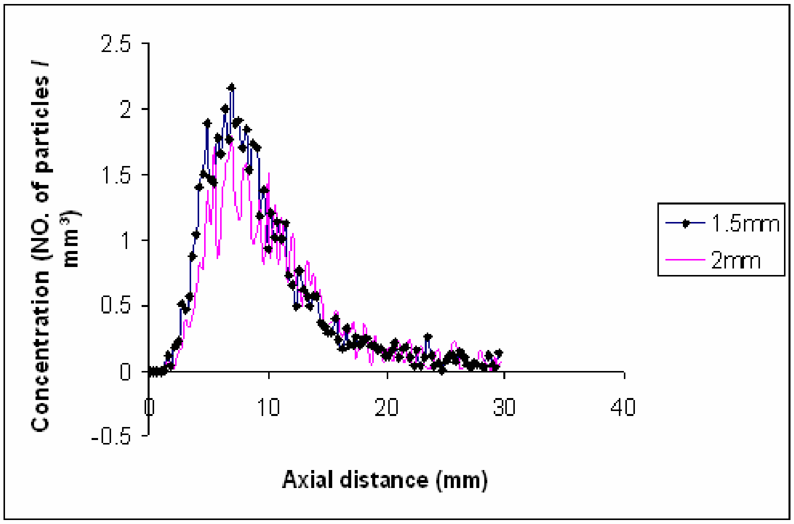
(a)



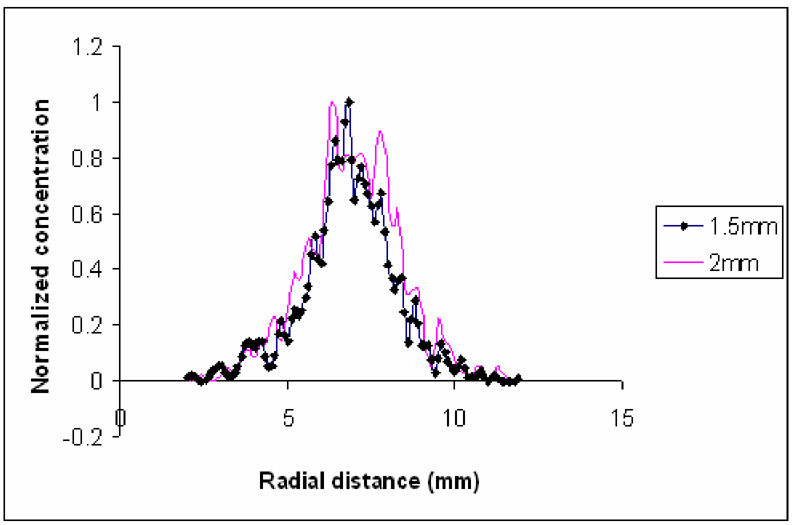
(b)

Fig. 4 Concentration variation in axial (a) and radial direction (b) for 45º and 60º nozzles.

Two nozzles with passage width at 2mm and 1.5mm are also evaluated numerically. Only the outer diameter of the coaxial nozzle is varied to obtain different opening width, as indicated by Fig.1. It shows the decrease of opening width leads to a slightly increase of concentration along central axis, shown in Fig. 5a. Fig. 5b shows that the profiles along radial direction are slightly different in two nozzles: the nozzle with 1.5mm opening width gives a narrower powder stream. The difference is very small, which can be attributed to the small difference in the opening width.



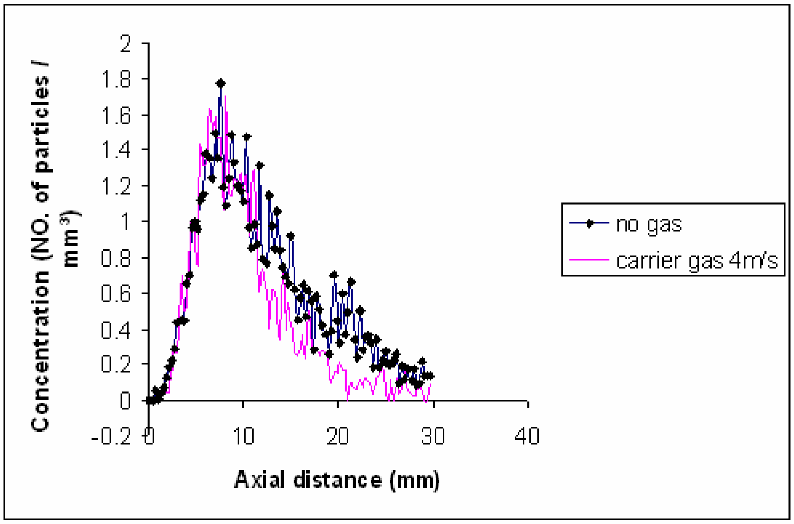
(a)



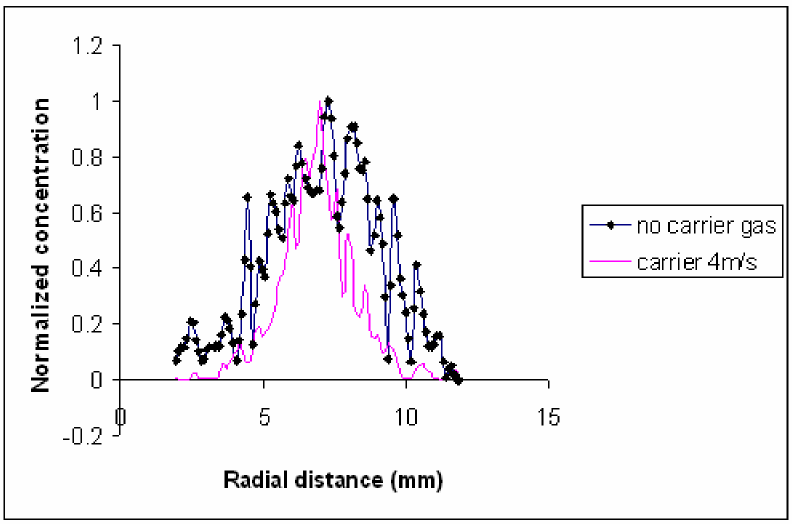
(b)

Fig. 5 Concentration variation in axial and radial direction for nozzles with different opening width.

The nozzle with carrier gas at 4m/s is then compared with the nozzle without carrier gas. The carrier gas is normally considered to suppress the dispersion of particles and bring particles to better focused stream. Due to the focus effect by carrier gas, the concentration at downstream when no carrier gas is applied is higher than that with carrier gas, as can be seen In Fig. 6a. The focusing effect can also be seen in Fig. 6b, where the diameter of focus spot with carrier gas is much smaller than that without carrier gas.



(a)



(b)

Fig. 6 Concentration variation in axial and radial direction for nozzles with different carrier gas.

CONCLUSION

In this paper, a numerical simulation scheme is described and used to evaluate main coaxial nozzle geometries and operational parameters. It is found that:

- ❖ Changing particle passage direction affects powder concentration along central axis greatly;
- ❖ Decreasing the particle passage width arise the powder concentration along axial and helps to decrease focus spot width.
- ❖ Using carrier gas can achieve a narrow powder stream.

These can be guidelines for future coaxial nozzle design. However further experimental studies have to be conducted to verify these findings.

ACKNOWLEDGMENTS

This research was supported by the National Science Foundation Grant Number DMI-9871185, the grants from the U.S. Army Aviation and Missile Command (AMCOM) and U.S. Air Force Research Laboratory contract # FA8650-04-C-5704. Their support is greatly appreciated.

Reference:

[1] J.Lin, W.M.Steen, Design characteristics and development of a nozzle for coaxial laser cladding, J. Laser Applications. Vol. 10, No. 2, pp. 55-63

[2] J. Lin, Laser attenuation of the focused powder streams in coaxial laser cladding, Journal of Laser Applications, Vol. 12, No. 1, (2000) pp. 28-33

[3] X. Yang, J. Wang, S. Gao, Y. Wang, Z. Zhang, Y. Tang, A new coaxial powder feeder without carrying gas for laser direct materials deposition, Preceeding of SPIE, Vol. 4915 (2002), pp.153-161

[4] A.J. Pinkerton, L. Li, Modeling powder concentration distribution from a coaxial deposition nozzle for laser-based rapid tooling, Journal of Manufcturing Science and Engineering, Vol.126, Feb. 2004, pp.33-41

[5] J.Lin, Numerical simulation of the focused powder streams in coaxial laser cladding, Journal of Materials Processing Technology, 105 (2000), pp.17-23

[6] H. Pan, Modeling powder flow for laser aided deposition process, Master's Thesis, University of Missouri – Rolla, (2004)

[7] B.E. Launder, D.B.Spalding, Lectures in mathematical models of turbulence, (1972)

[8] M. Sommerfeld, Modeling of particle-wall collisions in confined gas-particle flows, Int. J. Multiphase Flows 18 (1992) 905-926