Kinetic modeling of a high power fast-axial-flow CO_2 laser with computational fluid dynamics method

Qing Li (李 晴) and Youqing Wang (王又青)

National Engineering Research Center for Laser Processing, Institute of Optoelectronics Sciences and Engineering, Huazhong University of Science and Technology, Wuhan 430074

Received November 7, 2007

A new computational fluid dynamics (CFD) method for the simulation of fast-axial-flow CO_2 laser is developed. The model which is solved by CFD software uses a set of dynamic differential equations to describe the dynamic process in one discharge tube. The velocity, temperature, pressure and turbulence energy distributions in discharge passage are presented. There is a good agreement between the theoretical prediction and the experimental results. This result indicates that the parameters of the laser have significant effect on the flow distribution in the discharge passage. It is helpful to optimize the output of high power CO_2 laser by mastering its kinetic characteristics.

OCIS codes: 000.4430, 140.3470.
doi: 10.3788/COL20080607.0513.

High power fast-axial-flow CO_2 lasers with convective cooling have been extensively investigated experimentally and theoretically[1−9]. It is continuing to be a subject of research in purpose to modulate their operation. Particularly the characteristics of turbulent flow of laser mixture in high-power industrial CO_2 laser have an influence on energy parameters of laser beam. Grasps of the overall kinetics of the discharge process can be reliably used to at least predict upper limits of the laser's performance[2].

In order to describe and predict the laser mixture's dynamic characteristics, some numerical models for the analysis of the fast-axial-flow CO_2 laser were developed[1−9]. The much used approach to solve the models of the fast flow gas discharge is based on simultaneous solution of the rate equations and gas dynamics equations for the gas flow and light field in an optical cavity. Many researchers made a lot of contributions in this field. However, numerical solution of mathematical model is not precise enough because the precision of those calculating methods is limited. Computational fluid dynamics (CFD) method has become a powerful approach to analyze the three-dimensional (3D) turbulent flow in complicated domain[10]. CFD code used to solve the 3D Reynolds-averaged Navier-Stokes equations in a rotating cylindrical coordinate system for axial-flow pump can be used for the numerical modeling of the gas flow in fast flow CO_2 lasers.

The aim of this letter is to get a more accurate result to a 3D model of a 4-kW fast-axial-flow CO_2 laser by the CFD method. The model is constructed by a self-contained set of differential equations describing the flow and heat transfer and diffusion. The FLUENT software is chosen to solve the differential equations.

The following assumptions are used in the model:

1) Stimulated emission occurs only on transitions in the band 001 → 100.

2) The gas flow velocity is homogeneously distributed in the transverse to the flow direction, and the gas densities are dependent only on the distance along the axis of the discharge tube.

3) The gas properties such as temperature and pressure in the discharge tube are constant in time, but a linear function of the distance from the anode along the direction of flow.

4) The case is considered as a compressible flow and 3D geometry.

The computation grid is shown in Fig. 1. This laser is a fast axial flow laser equipped with 12 discharge tubes. The geometric model is a cylindrical discharge tube. The grid represents one of the discharge tubes of the laser cavity, whose size is about 320000 cells. The gas inlet and outlet are modeled as round openings which are located at each ends of the cylinder. We assume the laser cavity is axisymmetric. This is not valid in the vicinity of inlets and outlets but seems to be reasonable in the main part of the laser cavity. Other boundary conditions include the inlet static pressure, the total temperature, and the outlet static pressure[7].

The physical properties of the flow field are all the function of spatio-temporal coordinates[4]. In the present study, it is assumed that the active medium can be treated as a compressible ideal gas. The flowing process in discharge tube is described by the governing equations as follows.

Continuity equation:

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0. \quad (1)$$

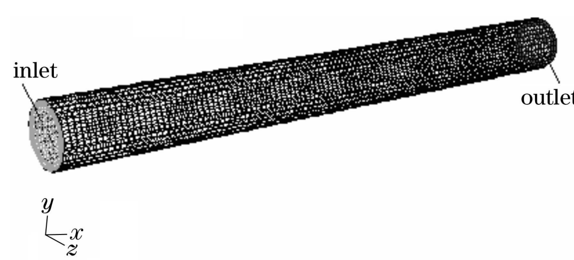

Fig. 1. A general view of the grid used in our computations from the side of inlet.

Momentum conservation equations:

$$\frac{\partial(\rho uu)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=-\frac{\partial p}{\partial x}+F_x, \quad (2)$$

$$\frac{\partial(\rho vu)}{\partial x}+\frac{\partial(\rho vv)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=-\frac{\partial p}{\partial y}+F_y, \quad (3)$$

$$\frac{\partial(\rho wu)}{\partial x}+\frac{\partial(\rho wv)}{\partial y}+\frac{\partial(\rho ww)}{\partial z}=-\frac{\partial p}{\partial z}+F_z. \quad (4)$$

Energy conservation equation:

$$\frac{\partial(\rho uT)}{\partial x}+\frac{\partial(\rho vT)}{\partial y}+\frac{\partial(\rho wT)}{\partial z}=\frac{\partial}{\partial x}\left(\frac{k}{c_\mathrm{p}}\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{k}{c_\mathrm{p}}\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial z}\left(\frac{k}{c_\mathrm{p}}\frac{\partial T}{\partial x}\right)+S_\mathrm{T}. \quad (5)$$

Gas state equation:

$$p=\rho RT. \quad (6)$$

In the above equations, ρ is the gas density, p is the gas pressure, T is the gas temperature, c_p is the heat capacity at constant pressure, k is the gas thermal conductivity, R is the gas constant, u, v, w are the gas velocities along the three directions of x, y, z. F_i $(i=x,y,z)$ giving friction losses on the three directions, and S_T is the dissipated heat per unit time and volume in the discharging process. Moreover, $S_\mathrm{T}=W_\mathrm{E}-W_\mathrm{L}$, where W_E is the input electric power per unit volume and W_L is the specific laser output power of one tube.

The FLUENT software uses a control volume-based technique to solve the transport equations (1)—(6). The differential equations are integrated about each control volume in order to convert them to algebraic equations which are solved numerically. The algebraic equations to be solved by FLUENT for any variable θ at point p may be written as

$$a_p\theta_p=\sum a_\mathrm{nb}\theta_\mathrm{nb}+s_\theta, \quad (7)$$

where the subscript nb denotes neighbour values, the coefficients a_p and a_nb contain convection and diffusion coefficients, and s_θ is the source term. The solution process is accomplished via a 'line-by-line' solver which gives rise to a tri-diagonal matrix, which is solved via Gaussian elimination. A flowchart of the CFD solving process is shown in Fig. 2. Computations are performed in steady state. The numerical results will be discussed afterwards.

The model takes into accout the standard k-ε turbulence modeling, that means the turbulence effects are considered. It is a semi-empirical model based on model transport equations for the turbulent kinetic energy k and its dissipation rate ε[10].

The laser consists of 12 discharge tubes with recirculating gas flow, which is provided by a turbo fan and two gas-to-water heat exchangers. The flow is a mixing gas of CO_2, N_2, and He at a mol proportion of 4.2:27.5:68.3. The material parameters such as molecular weight, thermal conductivity c_p, laminar viscosity μ, are computed with the corresponding proportion by different formulas[11]. As a compressible 3D flow model, the boundary conditions include the inlet static pressure of the gas inlet, the total pressure of the gas outlet, the operating pressure, the inlet temperature, the expected laser plasma static temperature, the fluid source energy items and other boundary conditions[7,10]. The input parameters of the laser model are shown in Table 1.

Computations based on the FLUENT software package allow us to obtian the distribution of the laser plasma parameters.

Figure 3 shows the variation of the velocity of the gas flow in the discharge tube. It can be seen that the gas velocity increases along the flowing direction, because a

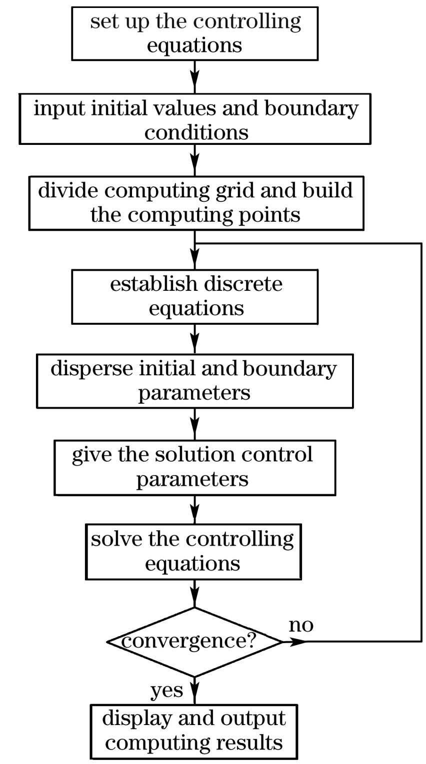

Fig. 2. Flowchart of the CFD solving process.

Table 1. Input Parameters of Laser and Boundary Conditions

Gas Composition	CO_2:N_2:He = 4.2:27.5:68.3
Molecular Weight (kg/m^3)	12.25
Thermal Conductivity, c_p (J·kg^{-1}·K^{-1})	1910
Laminar Viscosity, μ (kg·m^{-1}·s^{-1})	2.06×10^6
Average Operating Pressure (Pa)	13500
Average Gas Temperature, T_0 (K)	357
Discharge Tube Length, l (cm)	22
Voltage Drop across the Discharge (kV)	15
Diameter of Discharge Tube (mm)	19
Radius of Laser Beam (mm)	8
Gas Inlet Velocity (m/s)	206
Wall Heat Flux (W/m^2)	500
Fluid Energy Source, S_T (W/m^3)	9.1×10^6

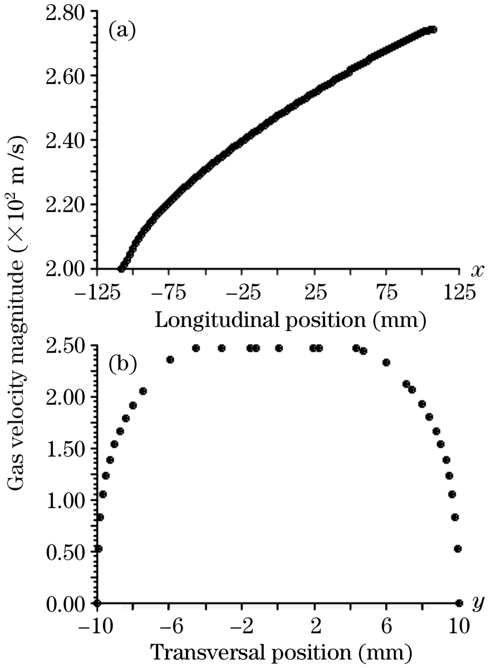

Fig. 3. Variation of the gas folw velocity in the discharge tube.

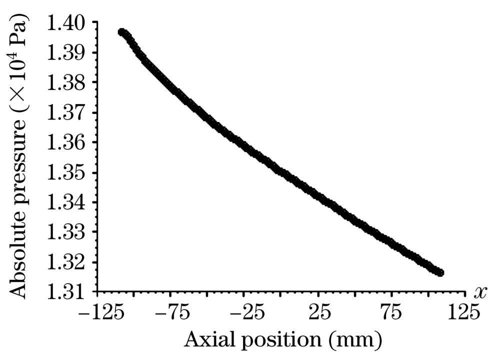

Fig. 4. Gas pressure along the discharge passage.

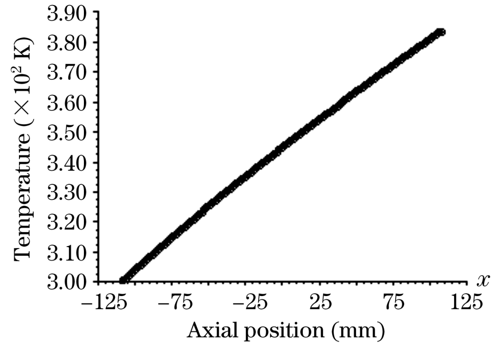

Fig. 5. Gas temperature along the discharge passage.

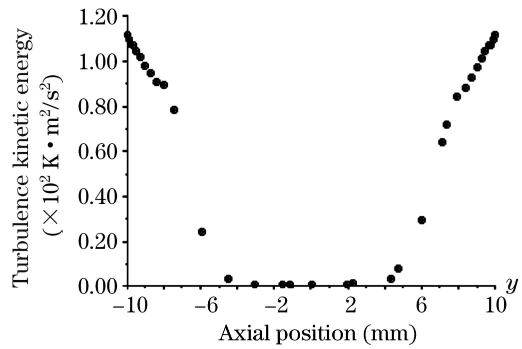

Fig. 6. Gas flow's turbulence kinetic energy along the vertical direction of the passage.

Table 2. Computed Experimental Results of the Gas Flow Characteristics

Characteristics	Experiments	Computed Results
Gas Mass Flux (kg/s)	0.00285	0.00285
Gas Flow Velocity (m/s)	208	209
Outlet Temperature (K)	428	425
Pressure Loss (Pa)	800	786
Laser Output Power (kW)	4.0	3.97

great deal of pumping energy injected in the tube converts into flowing energy. Moreover, the gas velocity near the wall comes down quickly along the vertical direction. At the position of the wall verge, the velocity magnitude is nearly zero due to the wall friction resistance.

Figure 4 shows a decrease of the gas pressure along the discharge passage. As a result of the high-speed and short gas flow, the loss of the pressure is even and small from the inlet to the outlet. The total pressure loss is less than 10^3 Pa.

Figure 5 shows an increase of the gas temperature along the discharge passage. Obviously the gas flow is heated up by the electrical pumping energy. As shown by the curve, the average temperature at the outlet is 385 K, a little lower than the practical gas temperature at the outlet of the glass tube. This is due to the high-speed flow; a great deal of the dissipated energy changes into fluid kinetic energy. Moreover, the temperature near the wall is much higher than that at the center.

Figure 6 presents the variation of the turbulence kinetic energy along the vertical direction of the passage. It can be seen that the energy distribution is symmetrical with the axis at center of the tube. It is corresponding with the velocity distribution in the same direction according to the hydrodynamics principles. Concretely the low-speed position near the wall is corresponding with a big turbulence kinetic energy, and the high-speed position in the center is corresponding with a small one.

The comparison of the computed results and the experiments of gas flow characteristics is given in Table 2. There is a good agreement between calculations and experiments whereas a slight difference exists. The difference is always being as the model is simplified, but that does not greatly affect the computed results.

The 3D turbulent flow in a fast-axial-flow laser discharge tube is studied by using CFD approach. A high-speed and low-pressure flow of the laser medium with turbulence in the discharge tube can be observed by numerical modeling. This result indicates that the initial conditions and parameters have significant effect on the flow distribution in the discharge passage. We can predict the flowing process on computer to achieve the optimized design instead of repeating tests and estimations.

The present study is preliminary. We believe that the reported model can be widely applied for quantitative modeling of the processes in industrial fast-axial-flow CO_2 lasers. The kinetic modeling of the CO_2 laser with CFD methods is worth to improve on. We should do more research on the interaction about the laser output and the gas flow characteristics, as taking into account the effect of the laser plasma's instability and coupling the vibrational energy equations of CO_2 and N_2 molecules, so that the vibrational temperatures will be predicted by numerical modeling.

Q. Li's e-mail address is vvlee83@126.com, and Y. Wang is the author to whom the correspondence should be addressed, his e-mail address is yqwang13@163.com.

References

1. S. Jelvani and H. Saeedi, Opt. Laser Technol. **40,** 459 (2008).
2. S. A. Hawat and K. A. Mutaib, Opt. Laser Technol. **39,** 610 (2007).
3. X. Peng, S. Li, and L. Xu, Proc. SPIE **2989,** 245 (1997).
4. D. Toebaert, P. Muys, and E. Desoppere, Infrared Phys. Technol. **38,** 337 (1997).
5. R. S. Galeev and A. A. Fedosov, Proc. SPIE **2713,** 8 (1996).
6. R. S. Galeev, Proc. SPIE **2117,** 205 (1994).
7. S. Sazhin, P. Wild, and C. Leys, Appl. Phys. **26,** 1872 (1993).
8. M. G. Baeva and P. A. Atanasov, Proc. SPIE **1810,** 109 (1992).
9. M. Stanghini, M. Basso, R. Genesio, A. Tesi, R. Meucci, and M. Ciofini, IEEE J. Quantum Electron. **32,** 1126 (1996).
10. F. Wang, Y. Li, G. Cong, W. Wang, and H. Wang, J. Hydrodynamics Ser. B **18,** 243 (2006).
11. P. Lu and R. Wang, Chinese J. Lasers (in Chinese) **28,** 775 (2001).