International Journal of Heat and Mass Transfer 65 (2013) 339–347

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Particle image velocimetry measurement of jet impingement in a cylindrical chamber with a heated rotating disk

Yao-Hsien Liu [a,*], Li-Wei Tseng [a], Chih-Yung Huang [b], Kung-Liang Lin [b], Chien-Chih Chen [b]

[a] *Department of Mechanical Engineering, National Chiao-Tung University, Hsinchu 30010, Taiwan*
[b] *Department of Mechanical and Systems Research Laboratories, Industrial Technology Research Institute, Hsinchu 31040, Taiwan*

ARTICLE INFO

Article history:
Received 21 February 2013
Received in revised form 15 May 2013
Accepted 9 June 2013
Available online 4 July 2013

Keywords:
Flow field
Particle image velocimetry
Jet impingement
Rotating disk
Chemical vapor deposition

ABSTRACT

We studied the flow field of a jet impingement on a rotating heated disk to simulate the flow field surrounding the rotating disk of a chemical vapor deposition (CVD) reactor, which is widely used for large-scale production of thin-films and semiconductor materials. The flow field influences the growth rate and deposition uniformity, and is subject to the combined effects of buoyancy, centrifugal, and flow inertia forces that occur during the deposition process. The study investigated various flow-cell sizes and locations, such as the inlet flow-rate (1–10 slpm), jet-to-disk temperature difference (40–80 °C), and disk rotational speeds (0–500 rpm). Particle image velocimetry (PIV) was used to measure the flow-velocity field and flow-streamlines in the test chamber. The time-averaged axial and radial velocity profiles near the disk were used to determine the variations in flow velocity resulting from rotation and heating. Upward buoyancy forces, caused by the heated disk, produce flow cells and break the flow uniformity above the disk. When the rotational Reynolds number increases, the rotational effect eventually dominates the flow field that increases the flow velocity and generates flow cells near the chamber wall. Flow regime maps of these flow patterns were constructed, based on the Grashof number and rotational Reynolds number.

1. Introduction

Chemical vapor deposition (CVD) is a widely used process for the manufacture of high-brightness LEDs, thin-film solar cells, and other optoelectronic components. Large-scale CVD production can be achieved by the deposition in a vertical cylindrical reactor. The deposition rate and the film-thickness uniformity are highly sensitive to the reaction chamber geometry, reaction pressure, inlet-flow flange design, wafer carrier temperature, and the carrier-gas flow rate. Inside the reactor, abrupt changes in the flow velocity or flow recirculation above the substrate are detrimental to the uniformity of the deposition. These deleterious effects can be minimized or eliminated by optimizing the reactor configuration and process parameters.

To improve the quality of CVD products, the fluid-flow behavior in the CVD reactor has been the subject of both theoretical and experimental research for more than two decades. Evans et al. [1,2] used numerical methods to describe the mixed convection behaviors of a rotating disk, and thus, simulated the vertical CVD reactor. Their simulation accounted for the buoyancy forces and for the effects of disk rotation, and revealed that non-uniform deposition can be improved by increasing the uniformity of the inlet flow and by increasing the substrate rotational speed. Joh and Evans [3] investigated the effects of varying the distance between the rotating disk and the inlet flow. Their 1D simulation showed that the heat transfer from the rotating disk becomes significantly more sensitive to the flow parameter and disk Reynolds number as the distance decreases. Also, the flow recirculation cell becomes smaller as the distance between the inlet and disk decreases. Weyburne and Ahem [4] tested the design and operation conditions in a water-cooled, close-spaced reactor that was used for the growth of III–V materials. The closer spacing between the injector and the susceptor leads to high utilization of the reactant gases as well as reduces residence time of the reactants on the substrate. Moreover, the convection flow cells can be suppressed to prevent non-uniform film deposition. Compared to a standard rotating-disk reactor, considerably better deposition efficiencies were achieved by close spacing, and they maintained excellent uniformity. Soong et al. [5] predicted the flow field in a rotating-disk metal-organic CVD reactor using numerical methods. They demonstrated that the epitaxial flatness can be tuned either by controlling the vortex under a rotationally dominant regime or by incorporating an inlet flow-control.

Flow measurement techniques have been applied to obtain the flow-field data from the CVD reactors. Horton and Peterson [6] used a Rayleigh light scattering system to measure the transient gas temperature in a simulated rapid CVD reactor. The flow field

---

[*] Corresponding author. Tel.: +886 35712121x55136.
E-mail address: yhliu@mail.nctu.edu.tw (Y.-H. Liu).

0017-9310/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.06.018

Nomenclature

D_j	diameter of the jet hole (m)
g	gravitational acceleration (m/s^2)
Gr	Grashof number ($=g\beta\Delta TH^3/\nu^2$)
H	distance between the jet hole and the disk (m)
N	number of sampling images
Q	inlet volume flow rate (slpm)
r, z	radial and axial coordinates
R	radius of the rotating disk (m)
Re	Reynolds number ($=\rho V_j D_j/\mu$)
Re_ω	rotational Reynolds number ($=\omega R^2/\nu$)
T_i	inlet fluid temperature (°C or K)
T_R	reference temperature ($=\frac{T_w+T_i}{2}$, °C or K)
T_w	disk temperature (°C or K)
ΔT	temperature difference between the disk and the inlet fluid (°C or K)
V_j	jet flow velocity (m/s)
V_r, V_z	radial and axial velocity components (m/s)
β	thermal expansion coefficient ($=1/T_R$)
ε	standard error of mean values
η	position error
μ	dynamic viscosity of fluid (N S/m^2)
ν	kinematic viscosity (m^2/s)
ρ	density of fluid (kg/m^3)
σ	standard deviation
ω	rotational speed (rad/s)

was dominated by the momentum before heating, but became unstable when the Gr/Re^2 ratio reached a value of 5. Mathews and Peterson [7] conducted flow-visualization and gas-temperature measurements to determine the regions of interest for momentum-dominated, buoyancy-dominated, and unstable flows. They later defined these regions as functions of the Grashof number, Reynolds number, pressure, and wafer temperature. Cheng et al. [8] studied the flow-field distribution in a CVD reactor using a flow-visualization smoke test, and investigated the convective vortex flow cells that were induced by the buoyancy and inertia forces by varying the pressure (300–760 Torr), jet-to-disk temperature differences (0–20 °C), and the inlet flow-rate (0.5–5 slpm). Their results revealed that lowering the chamber pressure reduces the buoyancy-driven circulation flows. They also pointed out that the variation of the air properties will be significant when the temperature difference between the jet and the heated disk is high. Setyawan et al. [9] investigated flow fields in a low-pressure (2.0–4.0 Torr) parallel plate CVD reactor because in high-temperature chambers, the particle trajectories are influenced by pressure. They showed that thermophoresis effects, resulting from the temperature gradient caused by heating the wafer-substrate plate, are pronounced for gas pressures of 2.0 and 4.0 Torr. Memon and Jaluria [10] experimentally investigated the flow structure and heat transfer in an impinging jet CVD reactor under atmospheric pressure. They investigated the momentum-driven and buoyancy-induced flow structures, and reported heat transfer correlations.

The reactor design has a significant impact on the flow-field distribution, and suppressing the buoyancy-induced flow recirculation can improve the epitaxial uniformity. Several techniques have been applied to reduce the flow recirculation, including inclining the reactor wall [11], tilting the cylinder head [12], and using rounded corners [13]. In addition, a uniform deposition is achievable by optimizing the processing parameters. Vanka et al. [14,15] numerically predicted the flow field in a CVD reactor; using an optimal inlet flow rate, substrate rotational rate, and reactor dimensionless length, the impinging jet reactor could be operated in atmospheric pressure without detrimental effects to the buoyancy-induced flow. Recently, Reinhold-López et al. [16] used particle image velocimerty to characterize the flow field in a vertically oriented cold wall reactor. The substrate surface temperature is up to 953 K and the gas flow rate ranged from 57 to 100 SCCM. Based on the results, the residence time curves and the minimum impingement time have been estimated.

Rotating-disk CVD reactors are popular for the production of large wafers because they offer better averages of the deposition distribution. The disk's rotation alters the buoyancy-induced and momentum-driven flows, which strongly influence the flow stability and uniformity of the deposition thickness. Biber et al. [17] showed that flow regime maps in a vertical rotating-disk reactor can be characterized as (1) plug-flow; the flow travels smoothly over the surface without causing any flow circulation in the reactor; (2) buoyancy-induced flow, in which an upward flow and recirculation forms during heating; and (3) rotationally induced flow, in which a toroidal vortex forms above the disk in the vicinity of the reactor wall. Plug-flow regimes are preferred for more uniform deposition. Kadinski et al. [18] numerically investigated the GaN/InGaN deposition in MOCVD vertical rotating-disk reactors. Their findings indicated that improvements in growth uniformity and alkyl efficiency are possible by modifying the alkyl injection system. Kim et al. [19] investigated the numbers of the injection holes and the rotating speed of the susceptor in a vertical RF-PECVD reactor. They concluded that the susceptor rotational speed has a significant effect, and that the buoyancy-induced flow should be prevented to provide greater efficiency and uniformity. Mitrovic et al. [20] investigated how flow stability is affected by a wide range of process parameters in the vertical rotating-disk MOCVD reactor, including the chamber pressure (10–1000 Torr), wafer rotational rate (0–1500 rpm), growth temperature (100–1100 °C), and the total gas flow rate (10–350 slpm). The flow regime can also be characterized by flow type as plug flow, buoyancy-induced flow, and rotation-induced flow. Flow recirculation due to the buoyancy force can be suppressed by increasing the total flow rate in the reactor or by decreasing the pressure. The literature contains reports of the optimization of the reactor design and process conditions [21,22]. Although numerical studies are readily available, detailed experimental reports of the effects of disk rotation on the flow field in a rotating-disk CVD reactor are scarce.

The objectives for this study were (1) to measure the flow velocity and flow streamlines using particle image velocimetry (PIV), which is more effective than traditional smoke visualization methods, because PIV provides spatial and temporal resolution; (2) the effects of the disk rotation and heating on the flow velocity above the substrate are investigated; and (3) a flow regime map is established to identify the plug flow, buoyancy-induced flow, and rotationally induced flow regimes in the test chamber.

2. Experimental setup and procedure

2.1. Particle image velocimetry system

Flow structures were investigated using PIV (TSI, MN, USA) (Fig. 1). The system comprises an Nd-YAG laser ($\lambda = 532$ nm) with a Q-switch module to control the pulsed laser energy to