Limit Design Costs with Fluid Flow Simulation

By Brian Drew,Business Business Unit Director,
Fluent Inc., Lebanon, NH, and

Lise Laurin, Founder/President of Clear Tech., Newton, NH

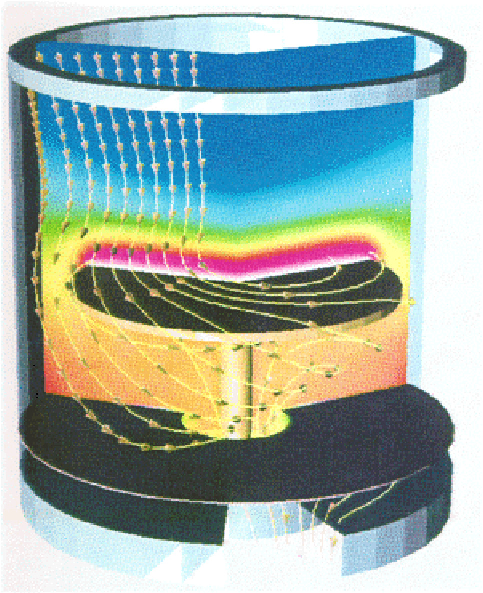

Figure 1. The 3-D model of initial MOCVD reactor conditions shows the sharp flow transition required for uniform film growth

New semiconductor equipment development has traditionally been funded bysemiconductor manufacturers who needed the new technology to move their own development forward. Increasingly, however, equipment manufacturers are being asked to fund equipment development on their own. At the same time, the cost to develop equipment has escalated to meet the demand for improved uniformities, lower particle counts, and larger wafer sizes. To reduce the cost of design, fluid flow simulation has become an increasingly popular tool, because it allows design engineers to test various process chamber configurations without having to build and physically test each one. Understanding flow patterns inside a processing chamber can lead to improved uniformities and fewer particles at a lower development cost.

Spin Coater Development

One such processing chamber is the bowl of a spin coater. The air flow distribution in a spin coater affects both the photoresist thickness uniformity and the number of particles deposited on the wafer. The final photoresist thickness depends upon the evaporation rate of the volatile solvent within the deposited resist. If the airflow across the wafer is not uniform, the resulting resist will be of a nonuniform thickness radially across the wafer. (The rapid spinning of the wafer confines nonuniformity to a radial pattern.)

During the spin-off process, the wafer is accelerated to very high speeds that propel excess resist from the wafer by centripetal forces. This process generateslarge numbers of resist particles, which, if not efficiently swept away, will likely wind up on a wafer at some point in time. Recirculation zones inside the chamber can cause these particles to be redeposited on the wafer. Any interruption of airflow through the bowl will cause the particles to be deposited on the walls of the chamber, where they create a hazard for future wafers.

Determining Optimum Flow Patterns

To determine the flow pattern inside a spin chamber, flow measurements can be made at several locations inside the bowl. Process uniformity and particle levels are also an indication of flow path. However, these methods are not enough to provide a thorough understanding of the complex patterns within the chamber. In the past, expensive trial-and-error methods were used to design these chambers, often resulting in less than ideal performance of the coating process.

Alternatively, computational fluid dynamic (CFD) modeling software provide significant insight into the flow patterns in even the most complex spin coating system. Models go well beyond simple flow measurements by allowing the design engineer to change conditions, such as exhaust flow rate, to determine the operating window of the coater. More importantly, models can predict flow patterns in conceptual chambers. A designer does not even have to build the chamber to determine its effectiveness. The designer can then optimize the conceptual chamber before any prototype is built.

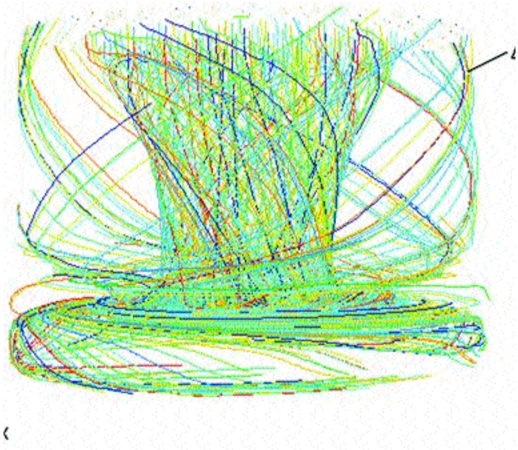

Figure 2. At 2500 rpm wafer spin speed and 250 fpm exhaust, air recirculation back to the top along the wall of the chamber poses a problem.

Case Study

X. Zhu and F. Liang at FSI International felt that CFD could aid their efforts to optimize spin chamber design. The first step in this process was to model an earlier version of the coater, the FSI Polaris ® Microlithography Cluster Coat Station. The engineers began by taking several hot-wire anemometer readings inside the chamber to gather flow information. The first measurements were at the top inlet to the chamber and showed a nonuniform velocity profile in this region. These data allowed the engineers to select a pressure boundary condition at this point instead of a velocity boundary. They then used a Dwyer manometer to measure the pressure at the exhaust ports of the chamber. Velocity calculations based on these pressure measurements correlated well with the actual measurements by the hot-wireanemometer.

Using the measured data as a starting point for modeling the chamber, the engineers entered this data into Fluent, a CFD software package from Fluent Inc. The chamber dimensions came directly from a CAD file and were entered automatically into the modeling software. The total amount of time required to set up the model was about 30 min.

The engineers began with a 2-D model, but quickly found that 2-dimensional modeling of the asymmetrical chamber geometry provided unrealistic results. A 3-D model of the chamber, on the other hand, provided results that appeared realistic and indicated a potential problem with the way the coater was set up.

Although the airflow coming into the chamber struck the wafer in the middle as expected, the majority of the air circulated back to the top along the wall of the chamber, instead of exiting through one of the exhaust ports (Figure 1).A deflector plate surrounds the wafer, so the air must flow between the deflector and the wafer to exit the chamber. A baffle sits slightly below the deflector plate. Pressure contour plots showed that the maximum pressure was underneath the deflector, between the deflector and the baffle. This high pressure area blocked the air flow from the wafer surface, so instead of flowing between the wafer edge and the deflector, the air was forced to return upwards along the walls of the chamber. The graphical results gave the engineers a clear picture of what was occurring. The spinning wafer draws air from the top opening and forces it outward radially. If the wafer speed is high, but the exhaust suction is

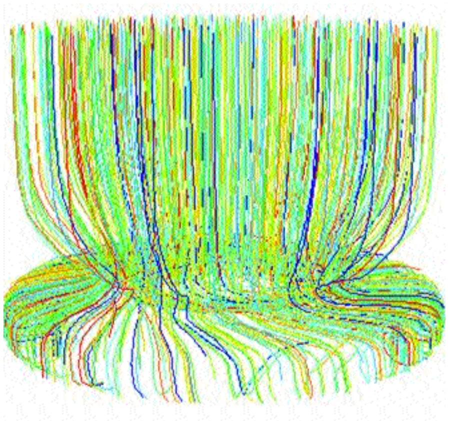

Figure 3. With an increased exhaust flow of 700 fpm and reduced spin speed of 1000 rpm, the air flows smoothly from the inlet to the exhaust.

low, too much air stays underneath the surrounding deflector, causing a high pressure area between the wafer edge and the exhaust duct. By increasing the spin speed in the model, the engineers were able to demonstrate that this effect worsened, strengthening the theory. The air velocities at the chamber inlet were much lower than expected. This indicated that the exhaust pressure was too low for this application. The resulting recirculation could be a cause of wafer contamination and therefore needed to be addressed. To test the theory further, the engineers modeled the system with higher exhaust flow (700 fpm). This greatly improved the airflow patterns and significantly reduced the positive pressure under the baffle. Reducing the wafer spin speed to 1000 rpm improved the flow patterns and completely eliminated the positive pressure under the deflector (Figure 2).

The model showed the engineers other critical information about this chamber configuration. In particular, there is little flow within 0.5 in. of the wafer surface in a circle about 6 in. in diameter about the centerline of the wafer. The top portion of the spin chamber has lower velocities and also has relatively uniform velocities when compared with the lower areas of the chamber.

These results gave the FSI engineers a clearer understanding of how the flow pattern is affected by the chamber geometry and the interactions between the spinning wafer, the chamber, and the exhaust flow rate. This provided a solid starting point for the engineers to begin design of their 300 mm Polaris Microlithography Cluster. The improved understanding of flow dynamics will enable the engineers to design a better system at significantly lower cost.

CVD Development

As in coating equipment, flow within a chemical vapor deposition (CVD) system plays a crucial role in both film uniformity and particle generation. The ability to model a chamber and understand the system dynamics without having to build and test a physical unit can save equipment manufacturers hundreds of thousands of dollars. The model can also show how the flow patterns are affected by changes in process conditions, such as gas flow rates, system pressure, and wafer temperature, giving the manufacturer a good idea of the full process window available in a specific design.

Scaling the Design

A common design task facing equipment designers is scaling the equipment for larger-sized or multiple wafers. It would seem that simply scaling the wafer chamber geometrically would maintain the material growth conditions over a larger area. In practice, however, geometric scaling produces less than desirable growth properties. By modeling the flow and reaction profile in the smaller chamber, the engineer designing the chamber can strive to emulate these profiles in a larger chamber, ensuring similar performance.

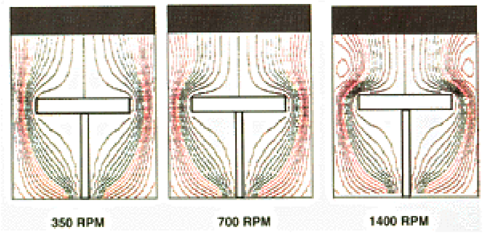

Figure 4. MOCVD flow patterns at three different spin speeds

As with resist processing, modeling a CVD chamber reduces the number of prototypes necessary to achieve a successful design. Testing alone can cost hundreds of thousands of dollars per prototype. By first modeling the system, prototyping is usually reduced to a single design, and testing of the design can be reduced to simple verification that the model is correct.

Case Study

Emcore Corp. uses CFD analysis software to scale its metal organic chemical vapor deposition (MOCVD) reactors. The company's proprietary TurboDisc deposition technology uses a high-speed rotating disc in a stainless steel chamber. The disc may hold one or several substrates. Controlled gases enter the chamber, where they react to form thin layers of compound semiconductors or advanced oxide materials on a wafer surface. As in spin coaters, the flow dynamics inside the chamber are affected by the rotating disc, as well as the inlet and boundary conditions. Emcore's family of systems relies on this technology, so each tool has similar design features and produces identical results. This means Emcore's customers can increase capacity with larger systems without new process development or training. For Emcore, scaling a system while maintaining process characteristics is a major selling feature.

Building and testing a single MOCVD reactor incurs significant expense. Beyond the cost of the equipment, running a larger production reactor costs as much as several thousand dollars per hour. Since it may take several months to characterize the machine, testing of each prototype costs the company hundreds of thousands of dollars. Emcore decided to minimize prototyping and testing costs by using fluid modeling to validate the design before beginning construction.

The unique design of the TurboDisc system provides uniform growth of III/V and II/VI materials. Reactant gases enter at the top of the chamber. The viscousforces caused by the rapidly spinning disc draw the gases down in a laminar flow pattern and then expel them outward across the disc surface. This flow pattern causes sharp and uniform temperature and gas composition gradients just above the disc surface, resulting in the thin, uniform boundary layer that is essential for uniform growth. If the geometric configuration of the chamber is not correct, the gases may recirculate above the disc, as in the spin coater example above. In this case, the gases coming off the disc have partially reacted, and include a mixture of bothunreacted gas and the byproducts of the reaction. As they recirculate, they dilute the incoming gas mixture, destroying the sharp boundary conditions at the surface.

To ensure that scaling the reactor did not cause recirculation zones, the Emcore engineers used Fluent software to model the reactor. Like the engineers at FSI, the Emcore engineers began with the characteristics of an older system. In particular, they chose the Reynolds number defining the viscous force of the spinning disc and a mixed convection parameter. To design the larger system, the engineers used these characteristics to calculate the ideal operating conditions of the new system, including disc size, rotation speed, chamber pressure, chamber diameter relative to disc diameter, chamber height, etc. Using Fluent, they modeled the resulting process to ensure the process was feasible and would produce the right gas flow patterns for good growth conditions For this project, the engineers used both 2- and 3-D models. Setting up the initial system conditions took about half an hour. Figure 3 shows the flow patterns under the initial conditions entered by the design engineers.

The next step evaluated the stability of the design under various process conditions. For this step, the engineers were able to use a 2-D model, saving computing time. Figure 4 shows the effects of spin speed on the gas flow. The graphic images produced by the modeling program clearly show the recirculation that occurs only when spin speeds reach 1400 rpm.

From these data, the engineers can set a maximum speed for optimum process conditions, or they can change the chamber design to eliminate recalculation in the desired process window. They can run the flow model on the revised chamber design to ensure it performs as required. When the design is finalized, the likelihood it will run to specification is much higher than if they had done no modeling. The few hours required to set up and run the model pay for themselves over and over in the reduced number of prototypes required and by limiting the number of tests required of the final prototype.

Emcore has used CFD software for the last four years, and it has played a major role in the design of their two newest systems, including the large 420 mm system.

Conclusion

Modeling software is playing a crucial role in limiting the ever-increasing cost of new equipment development. The advantages gained by a thorough understanding of flow patterns within the process environment include improved process uniformity, fewer particles, and rapid scale-up of processing equipment. Many semiconductor equipment manufacturers are turning to software like Fluent to improve the odds their equipment will perform to specification on the first prototype.