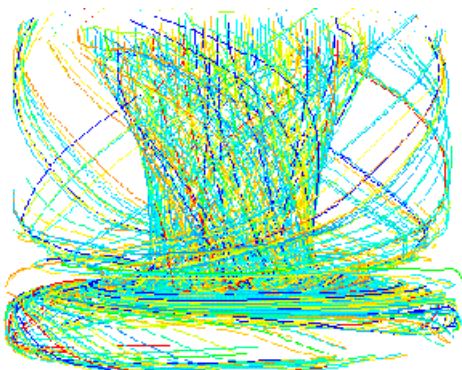


Fluid Flow Simulation Helps Improve Photoresist Uniformity

By X. Zhu and F. Liang
Microlithography Division, FSI International Corp., Allen, Texas



Air flow pathlines in the spin bowl at 2500 rpm wafer speed, and 250 fpm exhaust.

Fluid flow simulation helped to develop a new cluster spin bowl design that improved photoresist film uniformity and reduced particulate contamination of wafer surfaces. Designs were created for both 200mm and 300mm wafer diameters and optimized based on the modeling results over a range of operating conditions. These results were accomplished by developing a 3D model that predicted the flow field of the spin coater and made it possible to analyze different operating parameters and optimize the design of the spin bowl and exhaust system.

The air flow distribution in a spin coater has a critical effect on film thickness uniformity. The photoresist liquid contains a volatile solvent that evaporates during the spin coating process, and the final film thickness uniformity is determined by material spin-off and solvent evaporation. The solvent evaporation rate depends on the difference in partial pressures between the solvent in the air boundary layer next to the film surface and the bulk air over the surface, and

on convective transport of air flowing over the wafer. If air flow near the wafer surface is different from the ideal flow induced by an infinite spinning disk, then the mass transfer coefficient at the wafer surface will change with radial position and the resist film thickness will not be radially uniform.

Particle contamination

Airflow in the spin coater has a significant effect on the particle contamination of the wafer. Large numbers of resist particles are generated during the spin-off stage. If these particles are not immediately removed in the exhaust, they can be redeposited on the walls of the chamber. To prevent this from happening, it is important that airflow remains positive in a direction leading from the wafer to the exhaust of the spin coater without any recirculation zones that could redeposit particles on the wafer.

There is no practical experimental method of accurately and completely determining flow patterns within a spin coater. Velocity measurements can be made at a limited number of points but these measurements are not enough to gain an understanding of the complex patterns within the irregularly shaped chamber of a spin coater. Another problem with a purely experimental approach is the cost and time involved in constructing a new chamber in order to determine the effect of geometrical design changes. For this reason, until very recently spin coaters have been developed using a trial and error approach that made it very difficult to optimize the design.

Flow simulation

FSI engineers felt that computational fluid dynamics (CFD) technology had advanced to the point where it could be used to solve this design problem. A CFD simulation provides fluid velocity, pressure and species concentration values throughout the solution domain for problems with complex geometries and boundary conditions. In conducting the analysis, a researcher may change the geometry of the system or the boundary conditions such as inlet velocity, concentration, temperature, etc., and view the effect on fluid flow patterns or concentration distributions. CFD also can facilitate detailed parametric studies that can significantly reduce the amount of experimentation necessary to fully characterize a design and thus reduce design cycle time and cost.

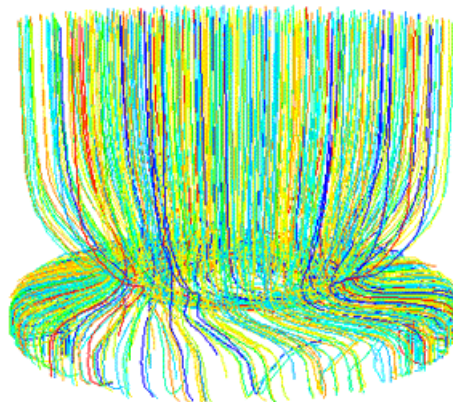
FSI selected FLUENT CFD software from Fluent Incorporated, Lebanon, New Hampshire because it handles the widest range of applications of any CFD code, including irregular geometries, radiation, multiple species, turbulent airflow and other complicating factors. The user interface makes it easy to define a simulation. Boundary conditions, material properties, turbulence models and particle tracking are specified through the use of tables and drop-down menus rather than by entering information on a command line. With most analyses, a user can be initiating an analysis in as little as 30 minutes after importing the analysis model or mesh.

2D and 3D models

Both 2D and 3D models were constructed for an older generation FSI POLARIS® microlithography cluster coat station. The air flow was assumed to be incompressible, turbulent and at a steady state. The 3D model was produced by importing the coat station

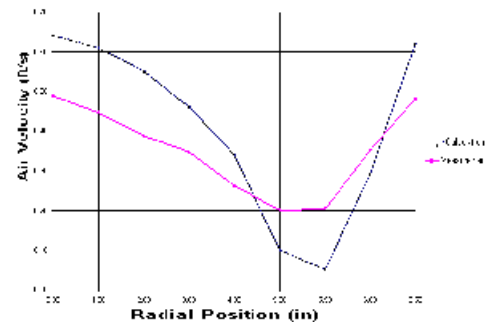
CAD database as an IGES file into FLUENT where the mesh was generated including all domains, wall cells and interior cells.

The 2D model consisted of 6,090 quadrilateral cells and the 3D model of 233,259 tetrahedral cells. The Renormalization Group (RNG) k-ε equation was used for the turbulent model and standard wall functions were chosen for the near wall treatment. In both 2D and 3D models, pressure boundary conditions were used at all inlet and outlet boundaries. The wafer surface was treated as a rotating wall.



Air flow pathlines in the spin bowl, at 1000 rpm wafer speed and 700 fpm exhaust.

A model calibration Dantec hot wire anemometer was used to measure air velocity across the top of the hood. The hot wire probe was mounted on a



Velocity magnitude profiles at hood inlet (wafer 3500 rpm, exhaust 250 fpm).

2-axis traversing device. These measurements showed that flow velocity is nonuniform across the top inlet so a pressure boundary rather than a velocity boundary condition was selected at the top inlet. Flow is very complicated near the exhaust suction ports, making it hard to define the velocity there, so static pressures at these ports were measured with flexible thin tubing connected to a Dwyer Manometer. The calculated velocity distribution compared favorably with the velocity profile measured by the hot wire anemometer. At 3500 rpm spin speed and 250 fpm exhaust, the average difference between the numerical and test results was 0.195 ft/s; at 2500 rpm and 250 fpm, it was 0.208 ft/s.

Both 2D and 3D models were analyzed under various operating conditions. The 2D model results were not

realistic given the non-axisymmetric nature of the coat station geometry. When analyzing the 3D model under identical conditions, an interesting phenomenon was observed. The air flow from the hood opening struck the wafer in the middle, some of it flowed to the exhaust but most of the flow recirculated back to the top along the wall of the hood. The exhaust mass flux was 0.0045 kg/s, of which 19% was from the top and 81% from the gap between the spin head and baffle. Pressure contour plots further showed that the maximum pressure was underneath the deflector and close to the exhaust channel. It was obvious that the air flow is blocked.

Backflow observed

The graphical CFD results provided engineers with a clear picture of what was occurring. When the wafer spins, it draws air from both the top opening and the bottom gap, and forces it outward radially. If the wafer speed is high but the exhaust suction is low, then too much air will stay underneath the deflector, from the edge of the wafer to the exhaust channel around the baffle. This results in a positive pressure buildup underneath the deflector that contributes to the flow blockage and also explains why there are no recirculation zones underneath the deflector as was widely presumed. The simulated backflow velocity at the edge of the deflector was 12.5 ft/s at 3500 rpm and 9.0 ft/s at 2500 rpm with the same exhaust setup. This demonstrated that higher spin speeds result in higher backflow. At the hood inlet, the velocities were seen to be generally much lower than the previously assumed 1.00 ft/s. In some cases, there was little or no air flow from the top inlet to the exhaust, indicating that exhaust suction was too low. Clearly the combination of backflow above the wafer and too much air drawn from the bottom gap posed a risk of wafer contamination.

Complex interactions

The analysis indicated that air flow in the coat station is affected by complex interactions of the spinning wafer, cup geometry, exhaust pressure and the gap between the spin head and surrounding baffle. For instance, when the operating conditions were changed to a higher exhaust setting of 700 fpm while leaving wafer speed at 2500 rpm, the backflow was significantly reduced. Under these conditions, the air

from the top inlet contributed 70% of the total exhaust mass flux and the air from the gap was reduced to 30%. Examinations of velocity vectors in the area near the wafer and the deflector at 2500 rpm wafer speed showed clearly that at 250 fpm exhaust most of the air flows back along the deflector edge while at 700 fpm most of the air goes into the exhaust. Furthermore, when conditions were changed to a 1000 rpm wafer speed and a 700 fpm exhaust, there was no backflow nor any positive pressure gradient at all.

The following general conclusions were drawn from these analyses:

- 1) There was little flow within a 6 inch diameter of the machine centerline on any level 0.5 inches or closer to the wafer. The profile in that area was flat and the overall velocity average was only 0.3 ft/s or less.
- 2) The velocity was highest in the vicinity of the annular channel between the edge of the wafer and the edge of the deflector opening.
- 3) Compared to lower levels, most of the top portion of the spin bowl had low velocities under 0.5 ft/s and relatively uniform velocity profiles.
- 4) Backflow could occur under certain operating conditions. Backflow was also observed in a flow visualization test.

The FLUENT results provided FSI engineers with a much clearer understanding of how the flow pattern in the spin bowl is affected by complex interactions between the spinning wafer, cup geometry, exhaust suction and the gap between the spin head and the surrounding baffle. They clearly saw that backflow of air could be generated under certain conditions, such as 2500 rpm and 3500 rpm wafer speeds with a 250 fpm exhaust. Based on the modeling and additional experimental results, the spin station geometry was modified for later generation tools. The new design improved the photoresist uniformity and minimized the particulate contamination of wafer surfaces. The CFD results also played a major role in the design of the new POLARIS® 300mm microlithography cluster.