Abstract Submitted
for the DFD15 Meeting of
The American Physical Society

Computational Study of Fluid Flow in a Rotational Chemical Vapor Deposition (CVD) Reactor SUN WONG, YOGESH JALURIA, Rutgers Univ — In a typical Chemical Vapor Deposition (CVD) reactor, the flow of the reacting gases is one of the most important considerations that must be precisely controlled in order to obtain desired film quality. In general, the fluids enter the reactor chamber, travel over to the heated substrate area, where chemical reactions lead to deposition, and then exit the chamber. However, the flow inside the reactor chamber is not that simple. It would often develop recirculation at various locations inside the reactor due to reactor geometry, flow conditions, buoyancy effects from temperature differences and rotational effects cause by the rotating substrate. This recirculation causes hot spots and affects the overall performance of the reactor. A recirculation fluid packet experiences a longer residence time inside the reactor and, thus, it heats up to higher temperatures causing unwanted chemical reactions and decomposition. It decreases the grow rate and uniformity on the substrate. A mathematical and computational model has been developed to help identify these unwanted hot spots occurring inside the CVD reactor. The model can help identify the user parameters needed to reduce the recirculation effects and better control the flow. Flow rates, pressures, rotational speeds and temperatures can all affect the severity of the recirculation within the reactor. The model can also help assist future designs as the geometry plays a big role in controlling fluid flow. The model and the results obtained are discussed in detail.

Sun Wong
Rutgers Univ

Date submitted: 01 Aug 2015 Electronic form version 1.4