Simulation of CVD Process in a Reactor

A. Kulkarni[1], F. Mumme[1], V. Frettlöh[1]

[1]Gemeinnützige KIMW Forschungs-GmbH, Lüdenscheid, NRW, Germany

Abstract

Metal-organic chemical vapour deposition (MOCVD) is a well-known process to produce high quality thin films that can protect, ennoble and functionalize metallic tools. To realize a high performance coating on a complex 3D mold surface, knowledge about the material formation from a gaseous phase and the penetration of the gas and the activated molecules into narrow contours and holes is necessary. For the adequate coating of a tool insert it would be very helpful to visualize the gas flow around the tool and into its cavities, so that the ideal position of the tool within the reactor can be determined without running time and resource consuming test series. Using COMSOL Multiphysics® software, the gas flow in the CVD reactor can be simulated and the gas velocities can be determined. The CFD module is used to carry out the simulations. As the process parameters used during practical experiments have a Reynolds number (1-2000), laminar flow is considered as the physics for the problem. By combining the simulation results and the knowledge of real coating experiments with diverse demonstrator tools a detailed picture of the gas and molecule pathways can be drawn and the trends in the observed layer thicknesses and coating flow lines on the samples can be explained. By doing so, the simulations not only help to realize an efficient coating on a tool surface within reduced time but also the amount of gas and precursor needed could be minimized. Moreover, the comprehension of the processes taking place during the CVD process is supported.

Figures used in the abstract

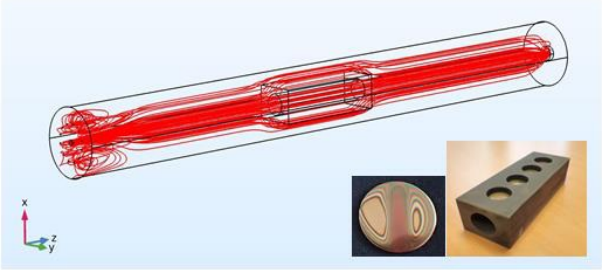

Figure 1: CVD Reactor, gemeinnützige KIMW Forschungs-GmbH, Lüdenscheid (Germany).