DESIGN OPTIMIZATION OF MOCVD REACTOR USING ANSYS FLUENT

By

Karthik Balasundaram

An Abstract of a Thesis submitted to the Graduate

Faculty of Rensselaer Polytechnic Institute

in Partial Fulfillment of the

Requirements for the Degree of

MASTER OF SCIENCE

Major Subject: ELECTRICAL ENGINEERING

The original of the complete thesis is on file
in the Rensselaer Polytechnic Institute Library.

Approved:

Prof. Ishwara Bhat

Thesis Adviser

Rensselaer Polytechnic Institute
Troy, New York
November 2010
(For Graduation December 2010)

ABSTRACT

Metal-Organic Chemical Vapor Deposition (MOCVD) plays a very important role in modern semiconductor processing. The design of the MOCVD reactor is very critical as it determines the quality and throughput of the processed semiconductor wafers. Since MOCVD processes are directly related to the thermodynamics and fluid mechanics of the flow of chemical precursors, the use of Computational Fluid Dynamics (CFD) based simulations for the study of gas flows appears to be very promising. In this thesis, we have tried to simulate the flow of gases in our horizontal MOCVD reactor using the CFD-based software FLUENT 12.0 from ANSYS. The work concentrates on analyzing the flow of gases inside the chamber walls of the reactor at conditions similar to the actual experiments done in our lab. We have tried to figure out if there are any gas recirculation problems in our present reactor using the simulations. Based on the initial results of our simulation, we have identified the presence of gas recirculation and tried to solve the problem by changing the geometry of the gas inlet tube. Through this thesis, we have proposed a new approach for MOCVD reactor design involving the use of CFD-based three-dimensional simulations of gas flows which can help in optimizing the design process for constructing or modifying any MOCVD reactor.