Dispersion of Particulate Matter in Air

Relevance to the Automotive Industry:	Computational Fluid Dynamics (CFD) is a vital tool for the automotive industry, due to the importance of fluid flows and heat transfer in automotive design and performance. Fluid flows and heat transfer occur within the engine (mixing of air and fuel in the engine and the subsequent combustion process; cooling of the engine; exhaust system and tailpipe emissions), external to the automobile (aerodynamic drag, handling, and wind noise), and inside the passenger compartment (heating and air conditioning), to name a few examples. Understanding and characterizing the behavior of fluid flows and heat transfer play an important role in increasing the performance of cars and can also help to face challenges arising from environmental concerns and limits of global fossil resources by increasing the overall car efficiency.	
Research Location:	TUD	VT
Homepage (Engl.):	http://www.graduate-school-ce.de	http://www.me.vt.edu
Faculty Mentor:	Prof. Dr. Michael Schäfer	Prof. Ken Ball, Ph.D.
Faculty Mentor Email:	schaefer@fnb.tu-darmstadt.de	ball@vt.edu
Graduate Mentor:	Dipl.-Math. Stephen Sachs Yu Du, M.Sc.	Sebastian Eluvathingal
Graduate Mentor Email:	sachs@gsc.tu-darmstadt.de duly@gsc.tu-darmstadt.de	sebastianje@vt.edu
Project Description: Jun 1 - Jul 29, 2009; (8 weeks, 40h/week)	Current research at Virginia Tech includes experimental and computational investigations of drag reduction in turbulent pipe flow, and also computational studies of fluid flows with particulate matter. Commercial CFD codes such as ANSYS-FLUENT can be used to simulate particle-laden flows with applications in the automotive industry. For example, tailpipe emission of particulates such as soot occurs due to the combustion process. The ability to predict the dispersion of particulate matter is essential to efforts to minimize the impact of such emissions in the environment. Continuing this research, the proposed REU summer students would be divided into two groups: - At VT: experiments with drag-reduced flows through rotating pipes using an existing rig; design and testing of a new rig to study the entrainment of particles in fluid flows for validation of numerical simulations; preliminary CFD studies of particulate flows; - At TUD: Perform numerical simulations to explore new active drag devices inside pipes; perform numerical simulations of particulate flows; validating CFD results with experiments. One student will be assigned to each research location. They will work closely together, staying in regular contact via email and video conferencing with each other and the project supervisors. Interaction is necessary to validate numerical results and to test drag reduction approaches discovered by computations. During the first two weeks, both students will go through an intense training phase consisting of an introduction to the principles of scientific work, individual literature review, brief overview on fluid dynamics and turbulence, and training for their respective research tools (experimental operations, data acquisition, CFD software, etc.). The last week of the summer research is reserved for making a presentation and writing a report in journal paper format.	
Necessary Skills/ Knowledge:	• Interest in learning about computational fluid dynamics	
Desirable Skills/ Knowledge:	•	
Additional Online Resource:		

NSF REU Students must have completed at least two semesters of engineering studies prior to the proposed summer research, and they must have at least one semester remaining before they can earn their BS in Engineering.