Control of particle agglomeration with relevance to after-treatment gas processes

KTH CCGEx

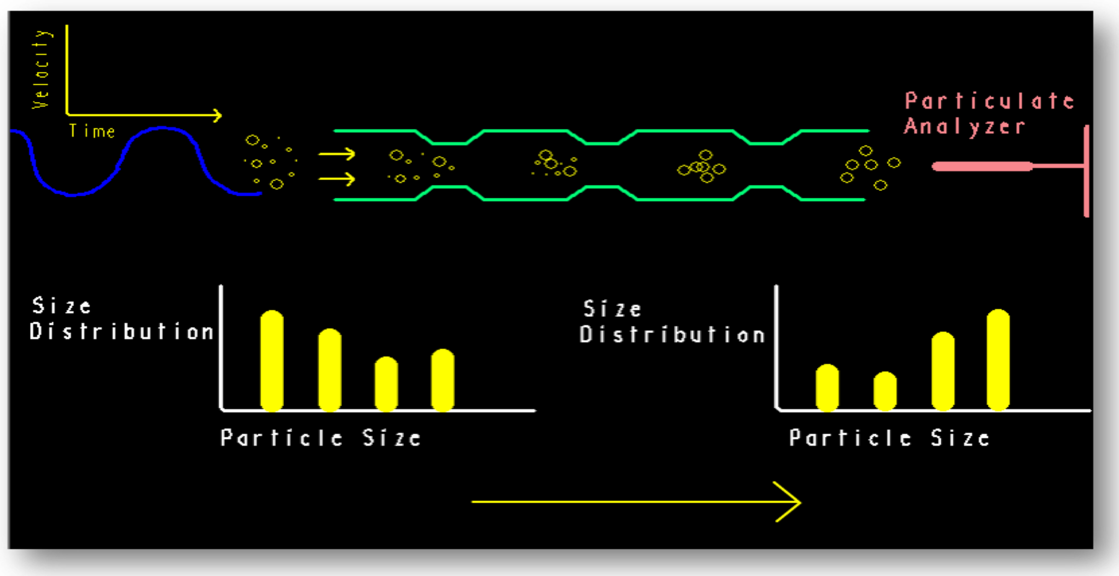
Ghulam Mustafa Majal

gmmajal@kth.se

The project aims to find methods for stimulating agglomeration of particles in the exhaust gases associated with internal combustion engines, by manipulation of hydrodynamic and acoustic fields. An understanding of the evolution of particulate characteristics (e.g. particle mass, particle number, size distribution) as traveling along the exhaust system components and connections with relevance to modern engines is needed. The enhancement of the aforementioned understanding will be developed with the usage of unsteady 3D CFD and possible experimental data from a parallel project. The flow is modelled using a Detached Eddy Simulation (DES) approach and the particle dynamics are modelled using a Lagrangian point force treatment.

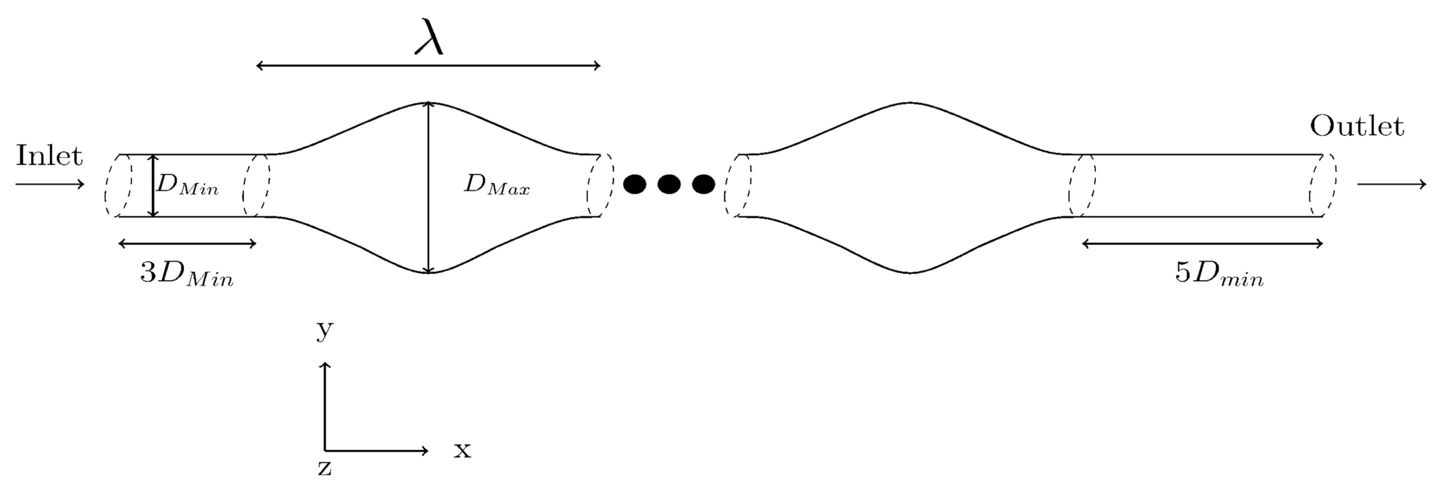
Introduction and Motivation:

Modern internal combustion engines tend to form fine particles prone to penetrate through the human respiratory system and cause cardiovascular as well as neurological problems. This project aims at deriving predictive models for the manipulation of particles in the exhaust line through the use of forced pulsatile flow conditions and/or acoustic fields.



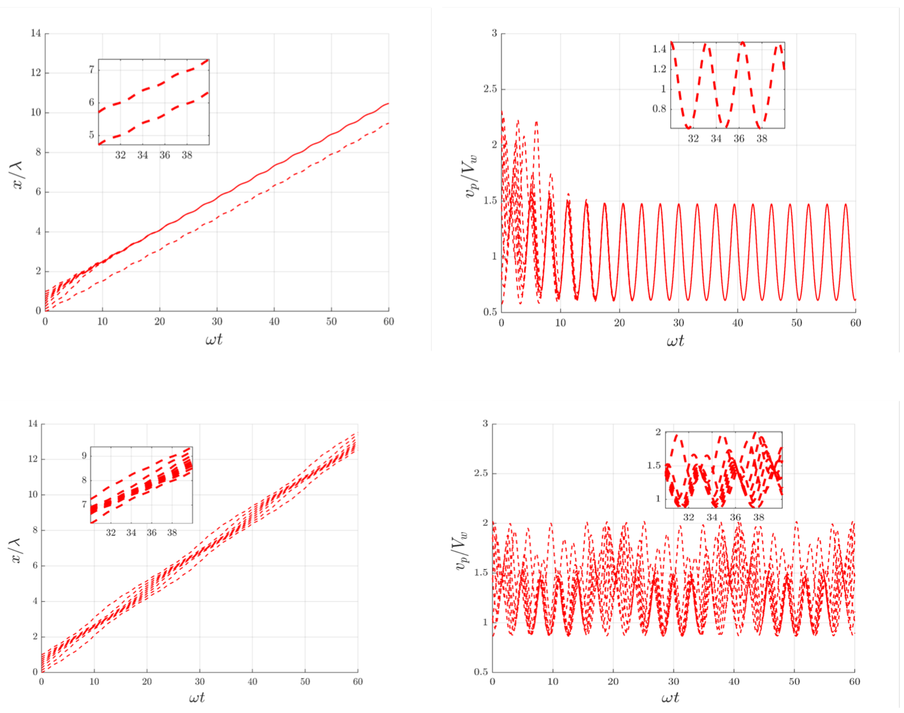
Setup:

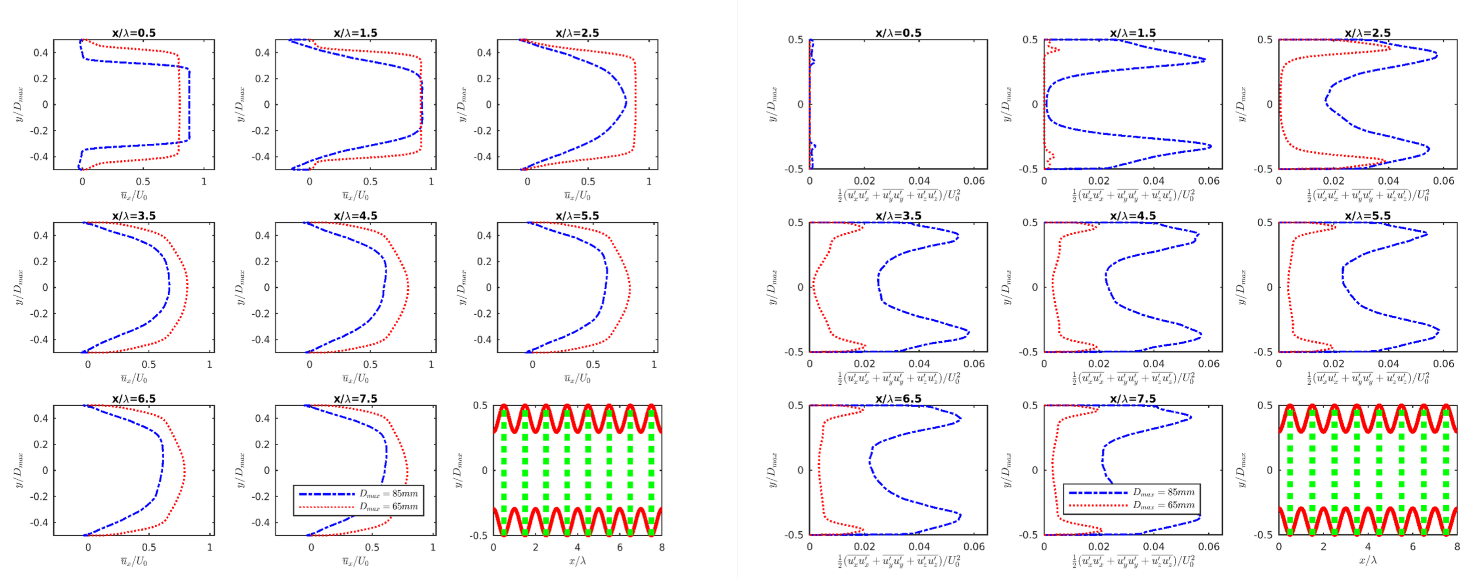
1D models for particle agglomeration process have been developed to analyze the effect of different flow and geometrical parameters on particle agglomeration. The model considered an oscillating Stokesian flow in a corrugated pipe. Only the drag force acting on the particle has been considered. Later, the simulations were extended to **3D** flow calculations using the Detached Eddy Simulation (DES) approach and applying a Lagrangian point force to account for particle dynamics. Non-pulsatile flow inlet conditions and a monodisperse distribution of particles were considered. Simple particle advection was considered.



Results:

In the leftmost figures (normalized) particle displacement and velocity for grouping (upper panel) and non-grouping (lower panel) scenarios using the **1D** modelling are shown. For the **3D** flow modelling a uniform velocity is imposed at the inlet of the corrugated pipe geometry. Two different geometries were considered having the same inlet diameter (D_{min}) and geometric wavelength (λ) but different maximum diameter (D_{max}). In the rightmost figures the normalized time-averaged streamwise velocity (left) and turbulent kinetic energy (right) are shown as predicted for the two different geometries (different D_{max}). The data is extracted along transversal radial lines located at different downstream positions in the corrugated pipe within the planes of maximum cross sectional area. A flow asymmetry is obvious as the D_{max} is increased.





Summary and Conclusion:

Tests have been carried out to understand the impact of the different non-dimensional parameters on particle grouping using a simple **1D model**. Conditions have been identified where particle agglomeration can occur in a corrugated pipe. **3D** flow scenarios have been studied for two geometries with the same inlet diameter and geometric wavelength but different maximum diameter (D_{max}). A flow asymmetry and higher turbulent kinetic energy levels characterized the set-up with larger D_{max} . Future work will include understanding the flow instabilities developed as increasing the D_{max} diameter, along with studying the impact on particle-particle interactions and particle distribution along the geometry.

Acknowledgement:

This project is supervised by Mihai Mihaescu (KTH-Mechanics), Mats Åbom (KTH-MWL), Lisa Prahl Wittberg (KTH-Mechanics), and Mikael Karlsson (KTH-MWL).

The support received from the Swedish Energy Agency, Scania AB, Volvo GTT, BorgWarner, and Volvo Cars is highly appreciated.

Part of simulations were performed on resources provided by the Swedish National Infrastructure for Computing (SNIC) at PDC Centre for High Performance Computing (PDC-HPC).