PROTECT
A COVID-19 National Core Study

Droplet modelling: understanding the physics of dispersal

This project uses computer modelling to develop a better understanding of the physics of droplet and aerosol behaviour that are key to the transmission of SAR-CoV-2, the virus that causes COVID-19.

Using Computational Fluid Dynamics (CFD) models, the project team are simulating the close-range indoor behaviour of aerosols and droplets emitted by talking and coughing, while also assessing the effect of control measures such as screens and ventilation. These models can be used to predict virus transmission and deposition patterns at various distances from someone either coughing or talking in a range of representative scenarios.

These CFD models will help us understand the effects of environmental factors and control measures on the risk of transmission of SARS-CoV-2. Examples of factors that will be assessed include different working environments, temperature and humidity (i.e. the risk of transmission in winter vs. summer). Control measures designed to reduce transmission, including physical distancing of people and separation screens, will also be assessed.

This analysis will provide insights about the factors that most affect transmission risk and the most effective control measures to reduce this risk.

The research questions the project aims to address include:

- What are the important fluid dynamics processes that affect the transport and dispersion of droplets and aerosols that are potentially carrying SARS-CoV-2?
- How effective are plastic screens in reducing the risk of transmission of SARS-CoV-2 if someone with the virus is coughing or talking?
- What effect does the ambient temperature and humidity have on the distance travelled by different size particles?

To address the questions, a detailed CFD model will be developed that takes into account the key physics of the transport and dispersion of droplets emitted when a person is talking or coughing. The model will represent the behaviour of the different size particles as they evaporate, travel through the air and are affected by environmental conditions such as ambient humidity and ventilation. The CFD model will be validated against experimental data for a person coughing, breathing and singing in a room.

Project team

Project lead: Dr Matthew Ivings, Health and Safety Executive

Other partners:

- Defence Science and Technology Laboratory
- University of Leeds

HSE [dstl] UNIVERSITY OF LEEDS