UNIVERSITÉ LIBRE DE BRUXELLES, UNIVERSITÉ D'EUROPE

Study on Aircraft Bleed Air Contamination by Engine Oil

Experimental and Numerical Investigation of Dispersed Air-Oil Two-Phase Flow through the Environmental Control System

M. Di Matteo, O. Berten, and P. Hendrick

Speaker: Mariano Di Matteo
Mariano.Di.Matteo@ulb.be

Aircraft Cabin Air
International Conference 2021
15-18 March 2021

Outline

Introduction

Methodology

Test results

Conclusions

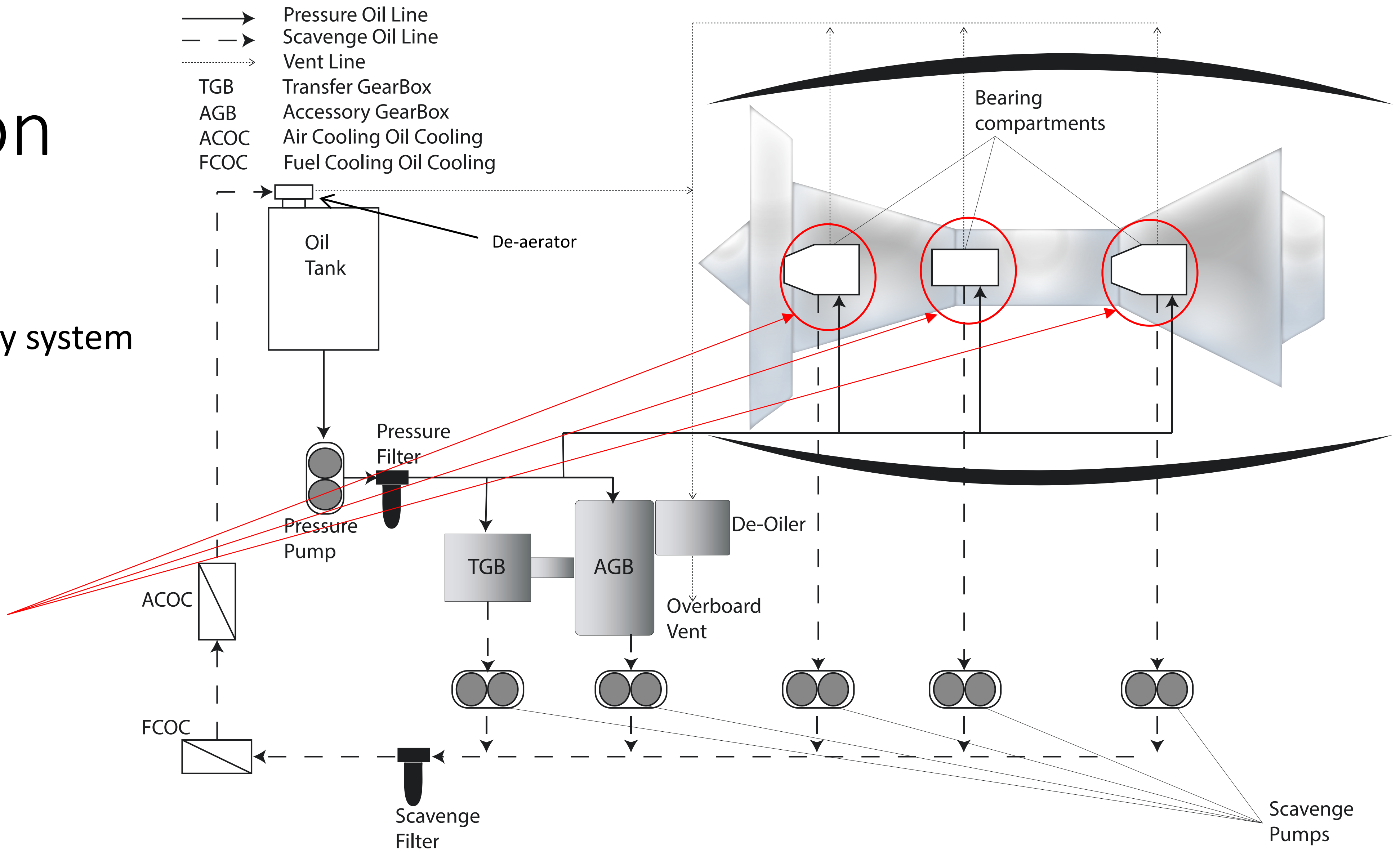
Introduction

- Environmental Control System supplies the cabin with fresh air at a certain pressure and temperature
- The air used by ECS is obtained by filtering and cooling the aero-engine compressor bleed air
- Bleed air is extracted from the compressor of the aero-engines or the APU
- A concern is the contamination of the bleed air by aero-engine oil

Introduction

Three oil subsystems:

- Storage and supply system
- Scavenge system
- Venting System



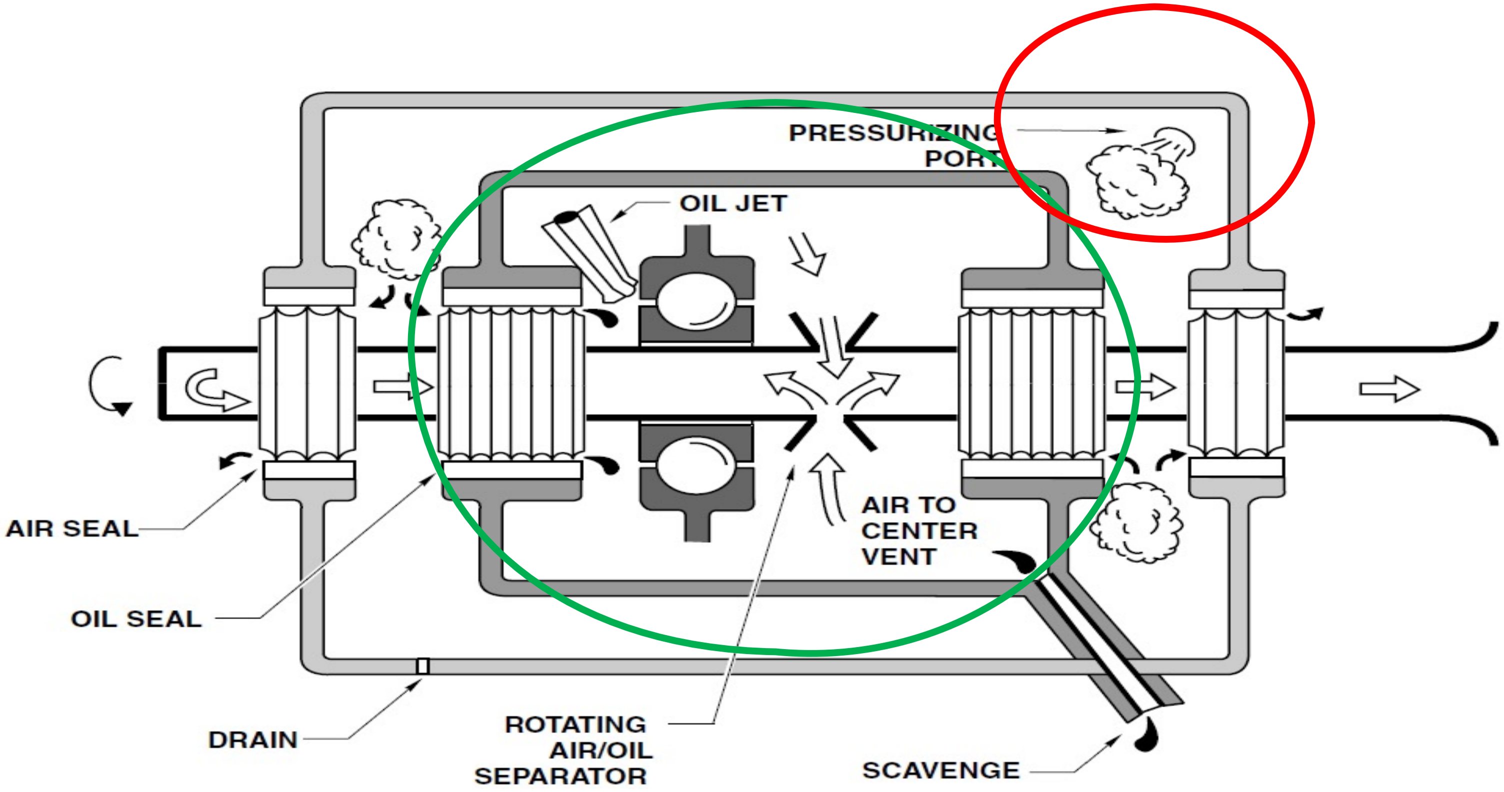
Failure at the level of the bearing chamber seals can lead to contamination

Introduction

Origin of the two-phase flow of oil and air

Aim of this study is to investigate the air-oil two-phase flow through the full piping system

Pressurizing port can become the problem during transients …



Introduction

There are three different exits for the air-oil two-phase flow:

Engine level:

- Pressurization pipe
- Vent line

Aircraft level:

- Environmental Control System

Methodology

Experimental phase:

- Droplet size measurement at inlet and outlet of piping system using the laser diffraction technique

Particle Measurement:

- Laser Helos-Vario/KR
- Measure droplet size from 0,5/0,9 to 175 µm (diameter)

Methodology

Experimental phase

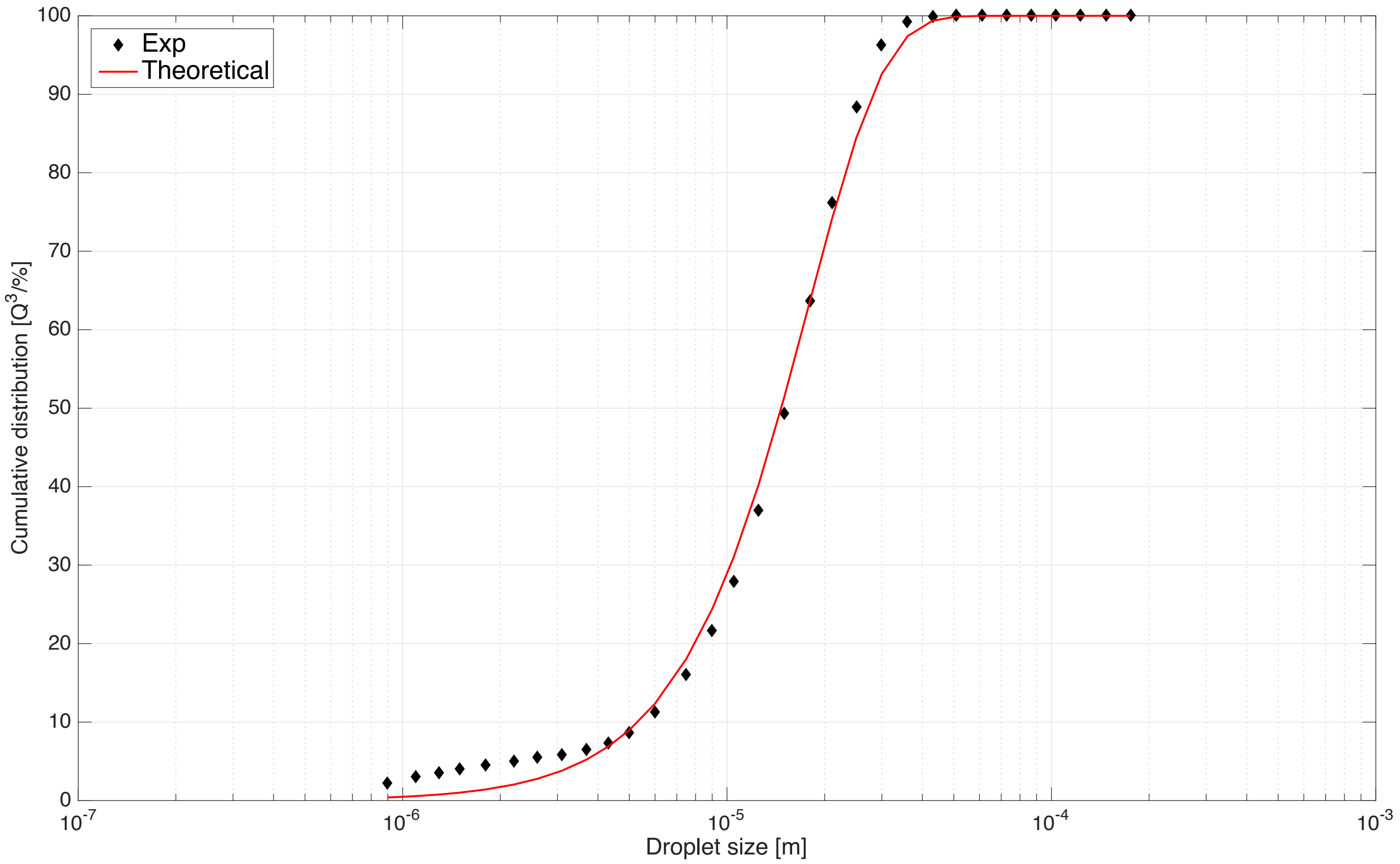
Piping system	Pipe diameter [mm]	Liquid (oil and water) flow rate [kg/s]	Air flow rate [g/s]
Straight 0.5 [m]	38 – 15 – 12	0.007 – 0.014 – 0.028 – 0.083	20 – 25
Straight 1 [m]	38 – 15 – 12	0.007 – 0.014 – 0.028 – 0.083	20 – 25
Straight 1.5 [m]	38 -15	0.028 – 0.083	20 – 25
Straight 2.5 [m]	38	0.028 – 0.083	20 – 25
Straight 3 [m]	38	0.028 – 0.083	20 – 25
Straight 3.5 [m]	38	0.028 – 0.083	20 – 25
Straight 5 [m]	38	0.028 – 0.083	20 – 25
1x curve 90˚	38	0.007 – 0.028 – 0.083	20 – 25
2x curves 90˚	38	0.028 – 0.083	20 – 25

Methodology

Experimental phase:

- Description of the granulometry using the Rosin-Rammler (RR) distribution

$$Y = 1 - exp\left(\frac{-x}{x_0}\right)^n$$



Methodology

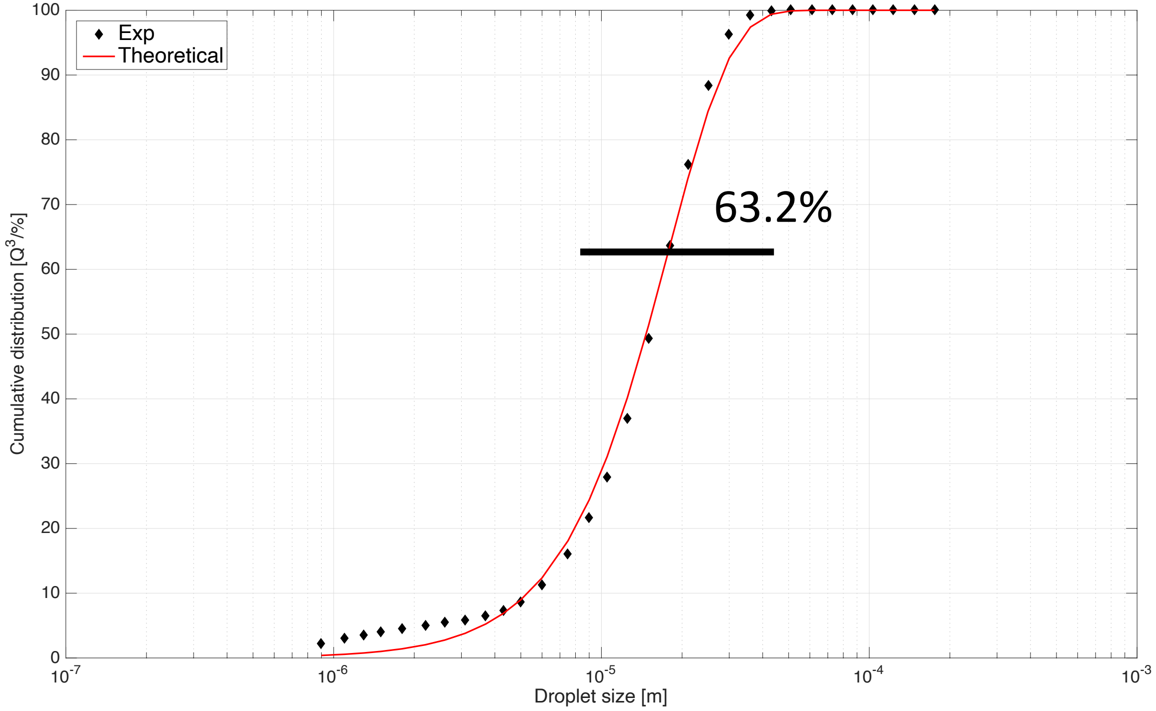
Experimental phase:

- Identification of distribution key parameter:
  - Mean diameter x_0
  - Spread parameter n

x_0 Corresponds to the diameter for which the 63.2% of particles are smaller

$$Y = 1 - exp(-1)^n = 0.632$$

$$n = ln\left(\frac{-\ln(Y)}{ln\left(\frac{x}{x_0}\right)}\right)$$



Methodology

Numerical Study

The inlet particle size distributions measured are used as boundary conditions for the numerical computations.

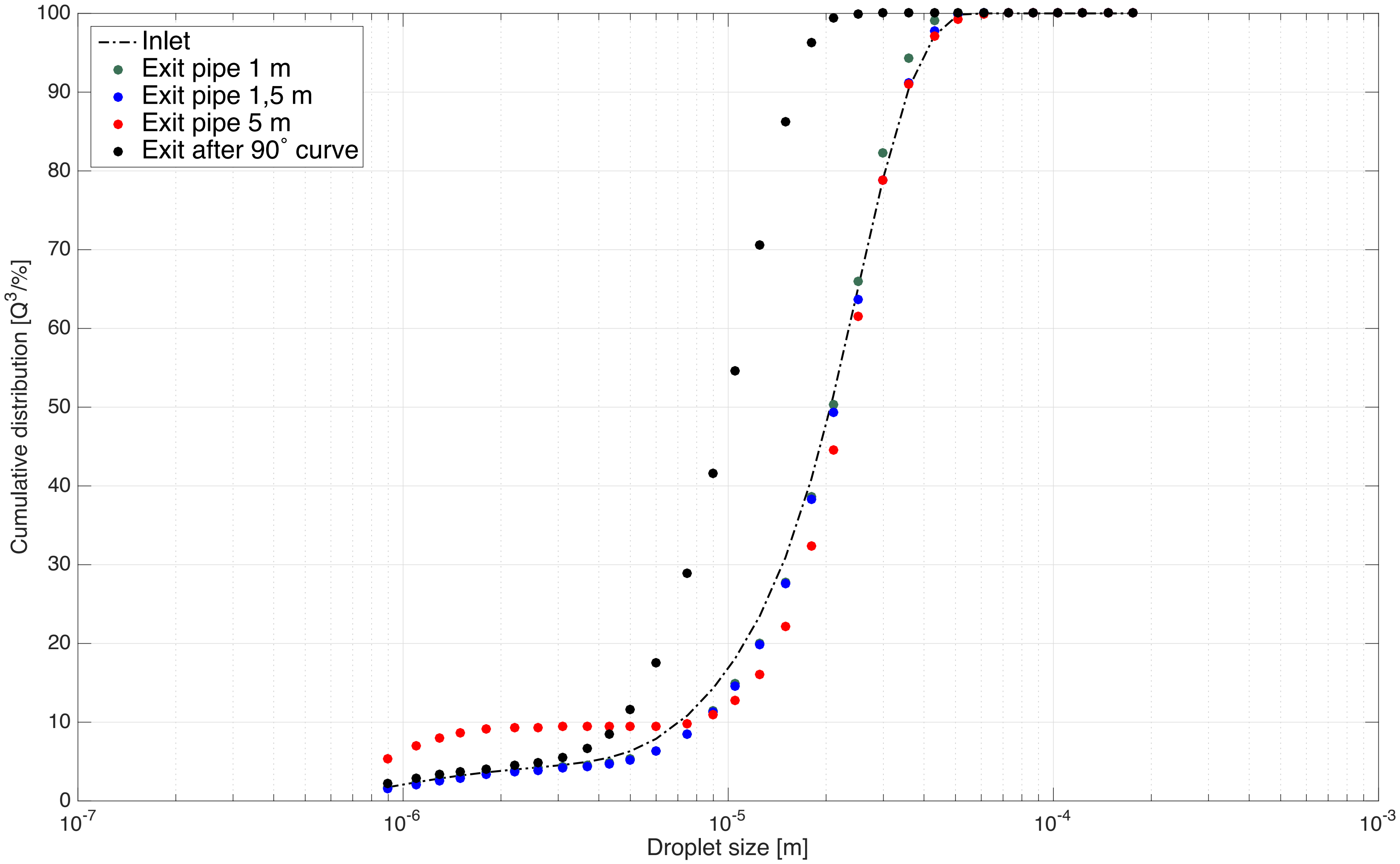
The outlet particle size distributions are used as validation data for the outcome of the CFD

Case	Straight pipe 0.5 [m] diameter 38 [mm]
Solver	Ansys Fluent
Turbulence model	$k - \varepsilon$ Realizable
Turbulence multiphase model	Dispersed
Discrete phase	DPM and DDPM
Inlet BC	Velocity magnitude of continuous phase
Discrete phase wall BC	Wall - film
Continuous phase time	Steady
Discrete phase time	Transient

Results

Experimental analysis:
The straight pipes slightly impact the droplet size

The presence of a 90˚ curve reduces the size of the droplets



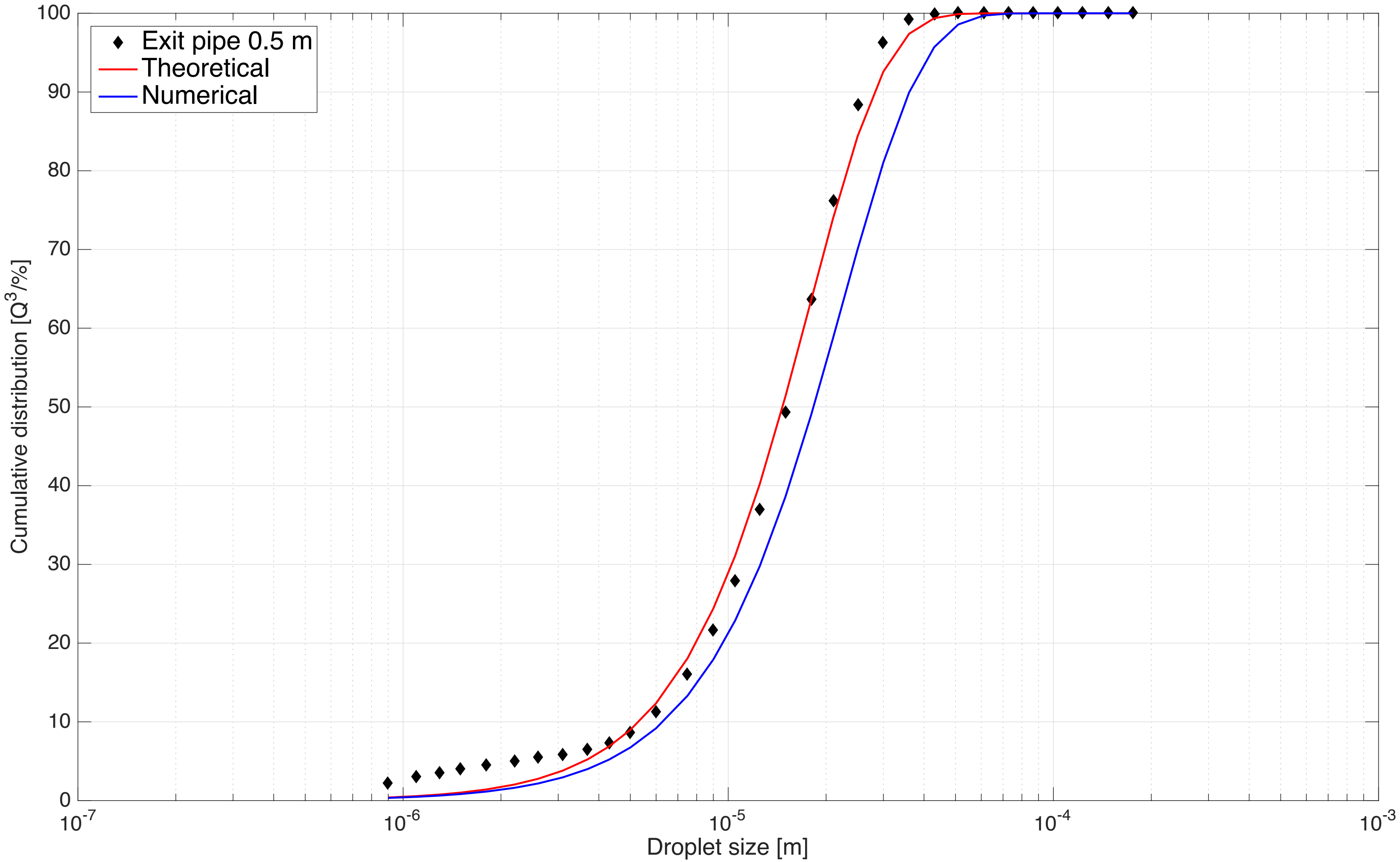
UNIVERSITÉ LIBRE DE BRUXELLES, UNIVERSITÉ D'EUROPE

Results

Experimental and numerical results at the exit of the 0.5m straight pipe.

The numerical curve is obtained by using the mean diameter and the spread parameter resulting from the CFD to define the RR distribution.

A good prediction is shown: The spread parameter presents a 5% difference with the experimental data.



Conclusions and Further Activities

Experimental capabilities outcomes:

- Straight pipe has slight impact on droplet size
- Presence of bends reduces the size of droplets
- The Rosin-Rammler distribution shows a good fit with the experimental results. This leads to the identification of the RR parameters x_o and n. Therefore, it is possible model the droplet distribution whit these parameters, and it can be used as BC and validation for the numerical analysis

Numerical outcomes:

- The experimental RR distribution is the BCs for the inlet of the DPM
- The potential of CFD in predicting the droplet size distributions is proven
- The ability to have a distribution at the exit of the pipe gives the possibility to predict droplet behavior for a complete circuit

Further activities:

- Simulation of a more complex circuit and explore different operating conditions
- Explore the possibility to predict in-flight conditions using simulations.