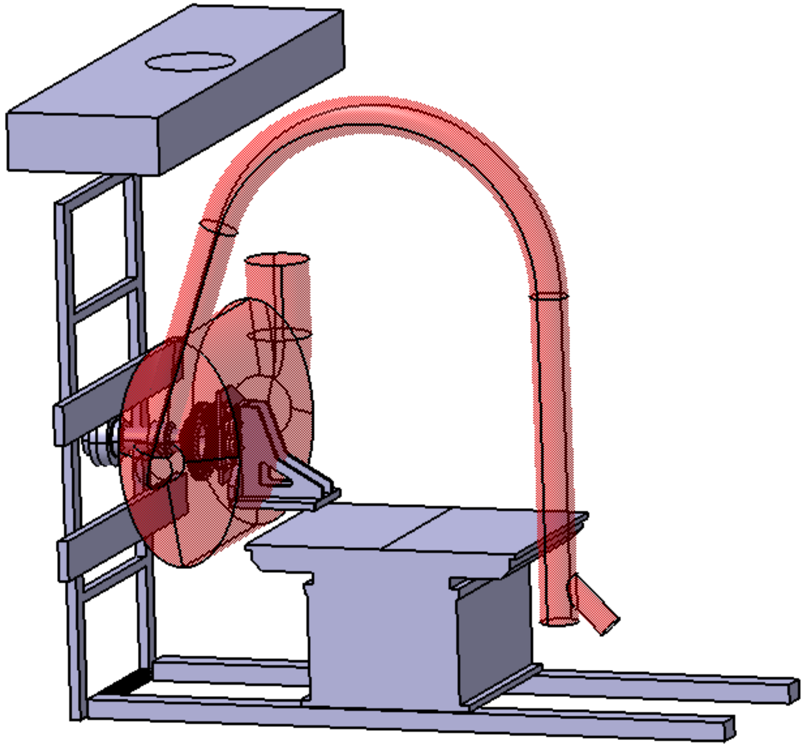
DESIGN OF A NEW PM DYNO-BENCH

Requirements

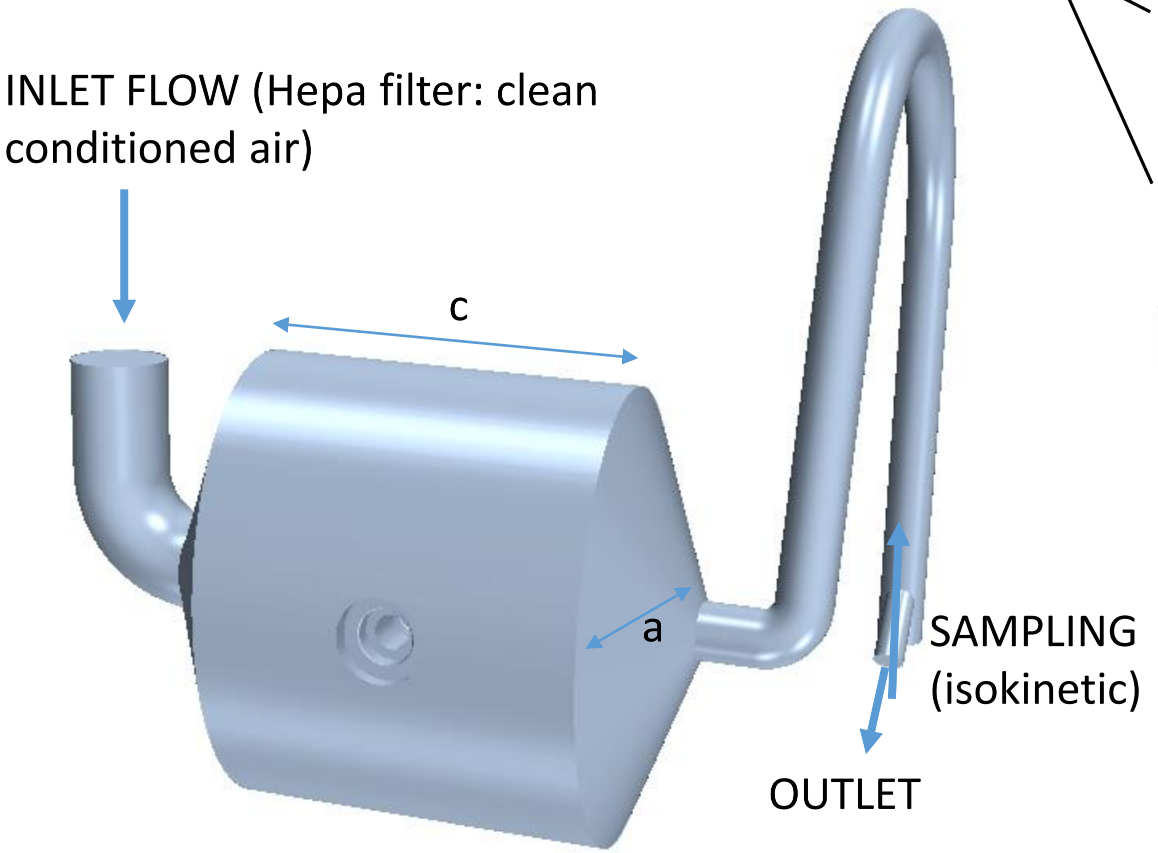
- Air-particle flow has to be well mixed
- The air flow hits the particle source (disc-pads interface) and goes directly to the outlet
- All the aerosol flow goes into the sampling pipe
- Avoiding angle decreases stagnation point where it is easier loosing particles.
- Isokinetic condition at the outlet

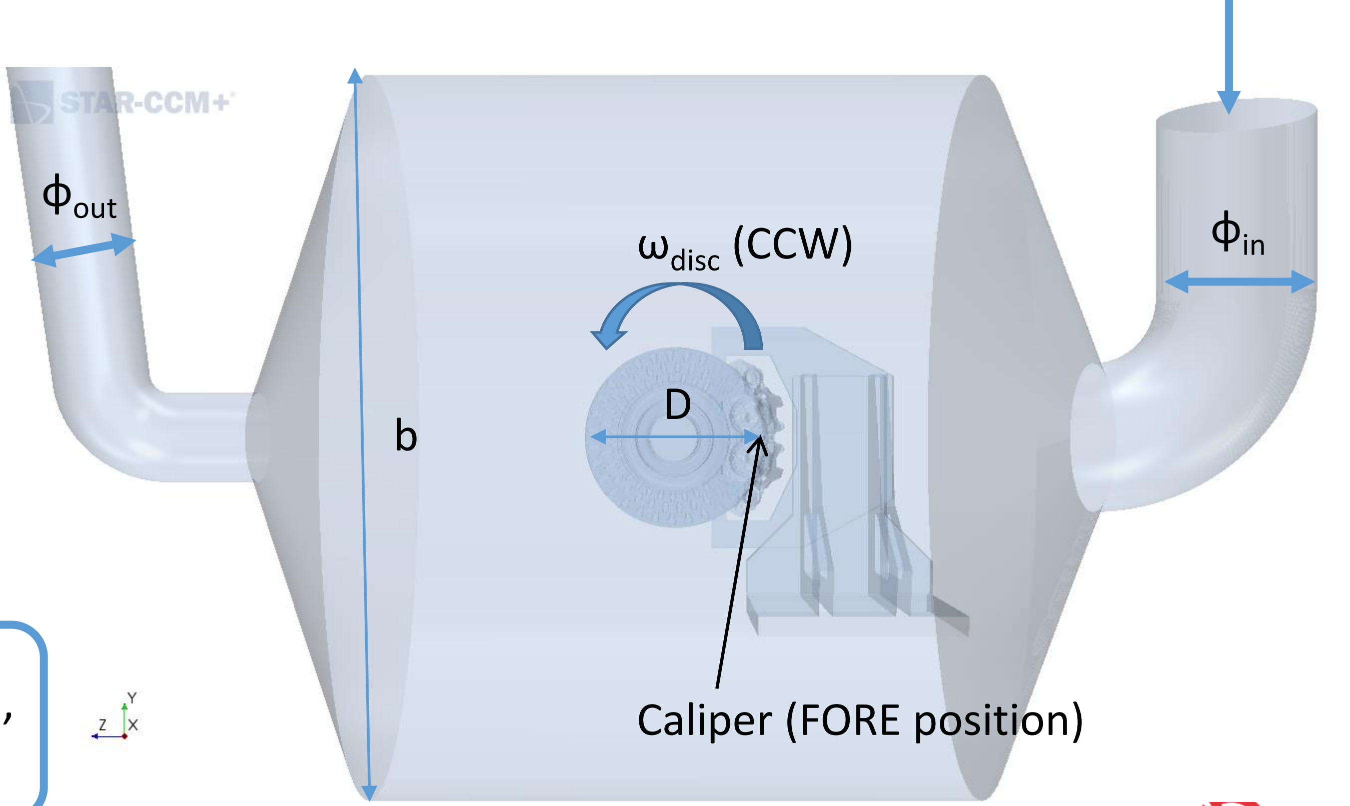


Dyno Bench Setup

Dyno bench configuration has been mainly based on:

- Straight developed flow at the outlet: long sampling pipe with defined diameter
- Minimization of particle losses: elliptical large room to avoid boundary effects, allow knuckle fixture option and let the disc to work properly from the ventilation point of view
- Design space (constrain): it has to be assembled inside a dyno bench





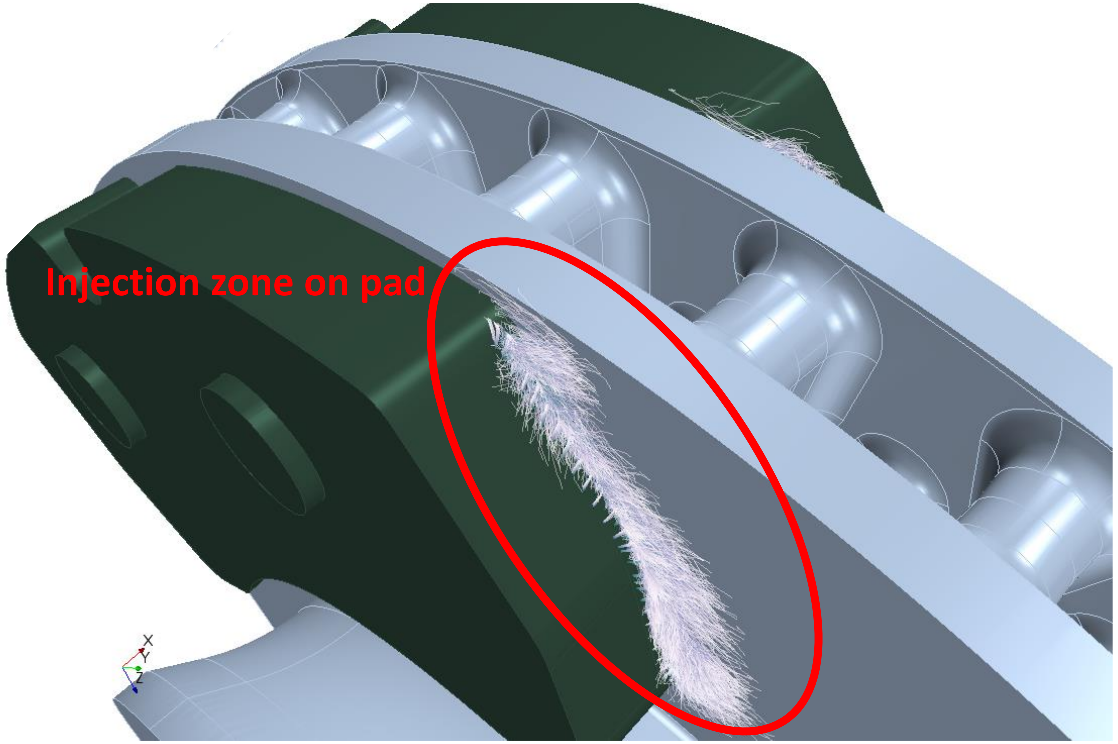
<u>Dimensions</u>: $c/b = 0.83$, $a/b = 0.67$, $D/b = 0.25$, $\phi_{in}/b = 0.21$, $\phi_{in}/\phi_{out} = 1.67$

Model Assessment

Standard CFD analysis has been coupled with a Multiphase Lagrangian approach to study the particles motion inside the new dust dyno bench.

Different particles diameter have been investigated:
0.1, 1 and 10 µm.

Particles are injected into the air from the side pad surface with a tangential velocity equal to *ωr.*



Preliminary work

First step of the work was the evaluation of the best model setup in order to guarantee the less number of particles losses as possible. Five different configurations have been investigated:

1) BASELINE: air pumped through inlet pipe and disc rotation counter clockwise

2) INVERSE DISC ANGULAR VELOCITY: air pumped through inlet pipe and disc rotation clockwise

3) AIR SUCKED FROM OUTLET (versus air pumped in the inlet)

4) 10% REDUCTION OF MASS FLOW INLET (low ventilation condition)

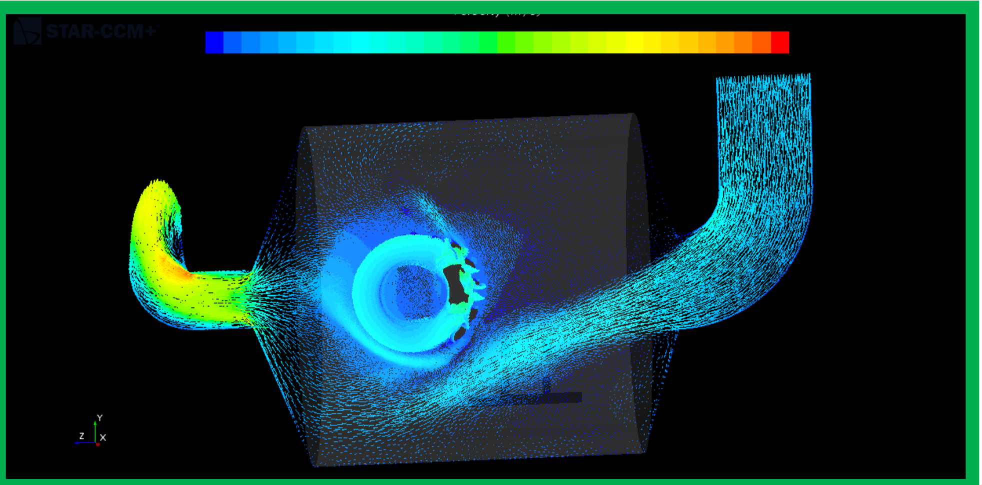
5) OUTLET ROTATED BY 180°

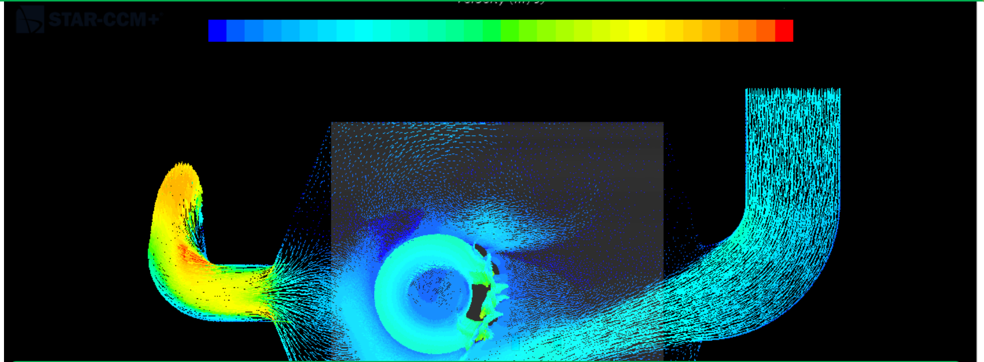
Simulations have been performed considering particles diameter of 10 µm, 1 µm and 10 nm

Model Assessment

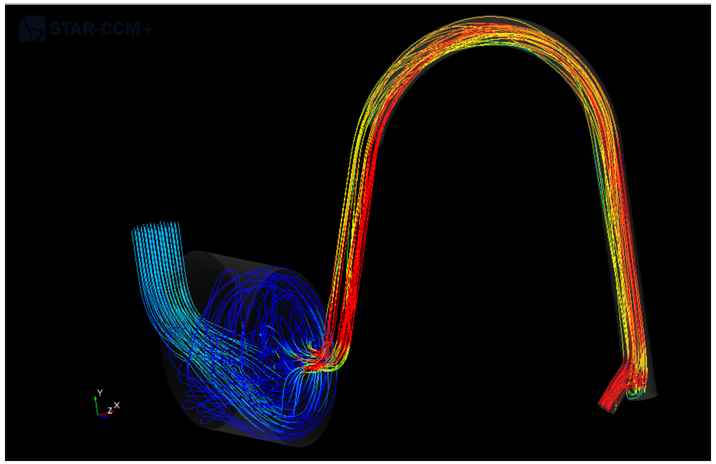
STEP 1. CFD Steady Simulation

The starting point of the method is the CFD analysis of the mono-phase aerodynamic flow inside the dyno bench, the disc rotating with no injection of particles.

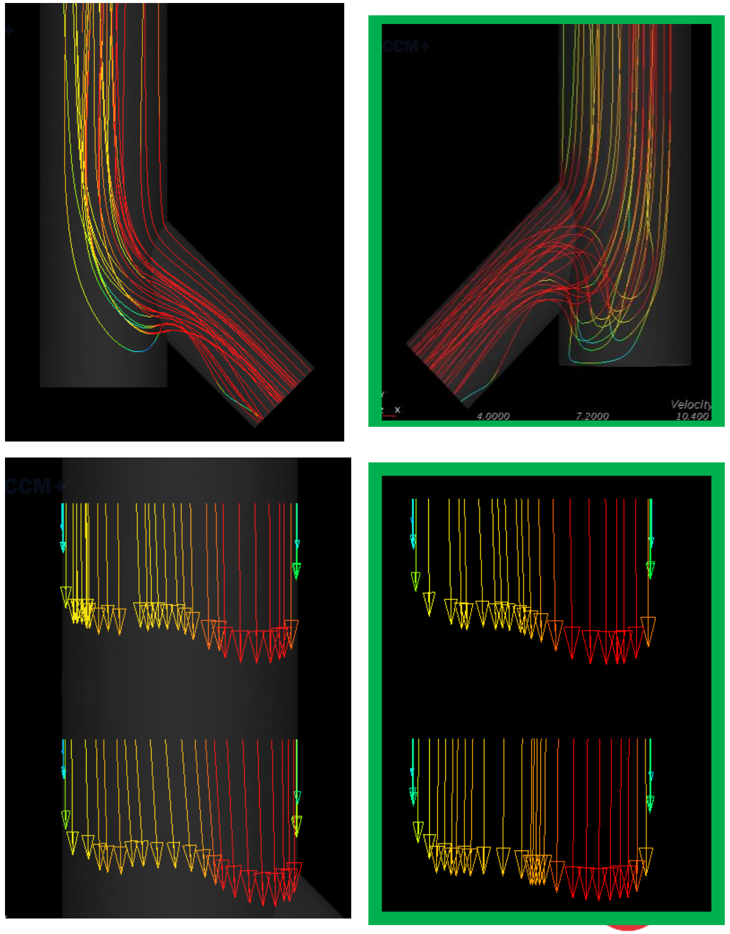




DISC ROTATION: CLOCKWISE INCREASES NUMBER OF IMPINGEMENT ZONES ON THE WALL CHAMBER



DISC ROTATION: COUNTER CLOCKWISE AIR PUMPED FROM INLET

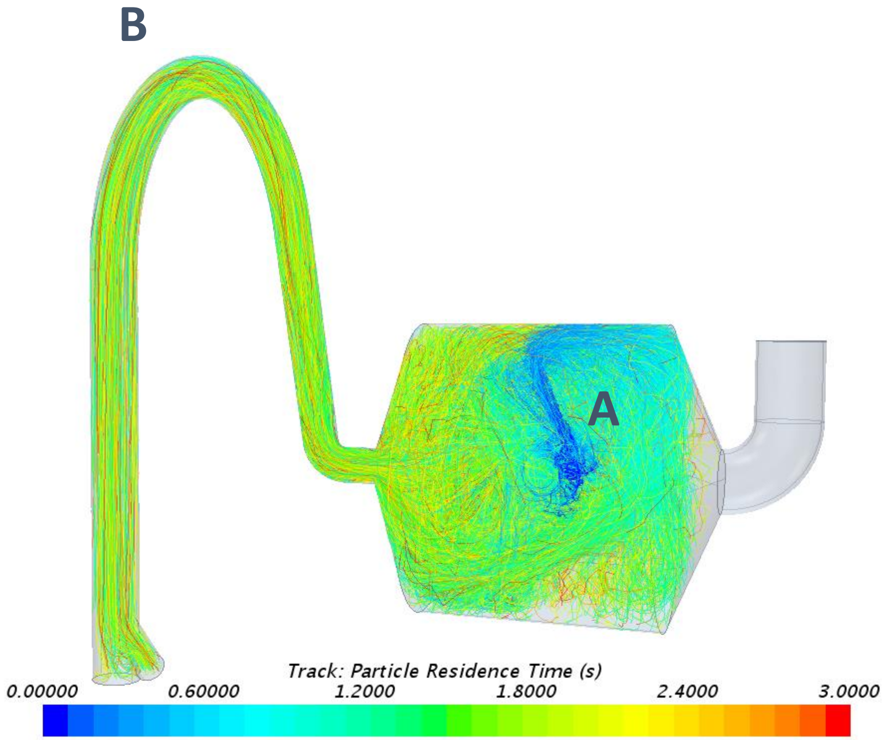


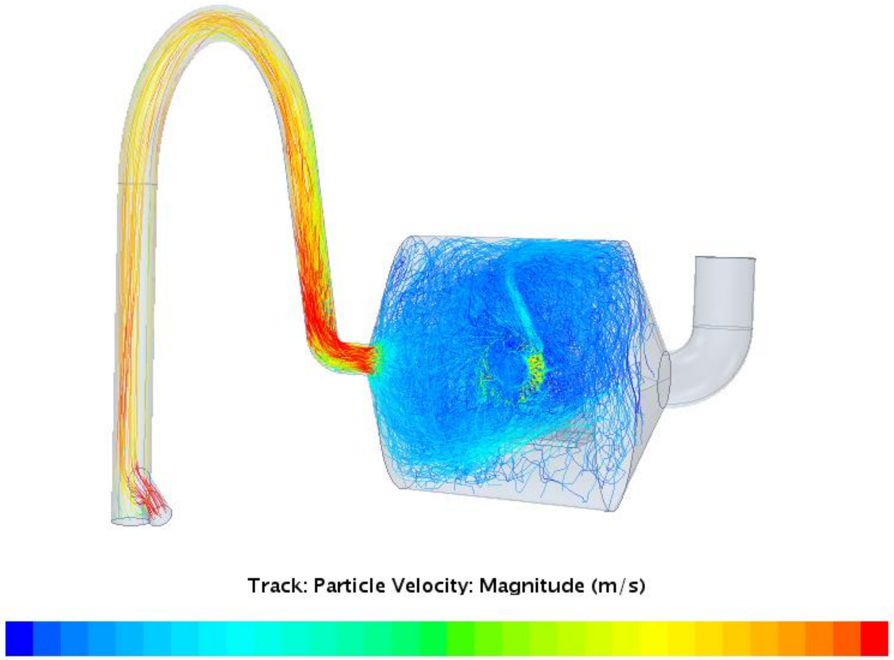
REDUCING MASS FLOW INLET REDUCES LOSSES
ROTATING EXIT BY 180° MAKES VELOCITY STRAIGHTER

Particle Losses Evaluation

STEP 2. Dispersed Phase Simulation

Evaluation of particles tracks by means of turbulent dispersion model





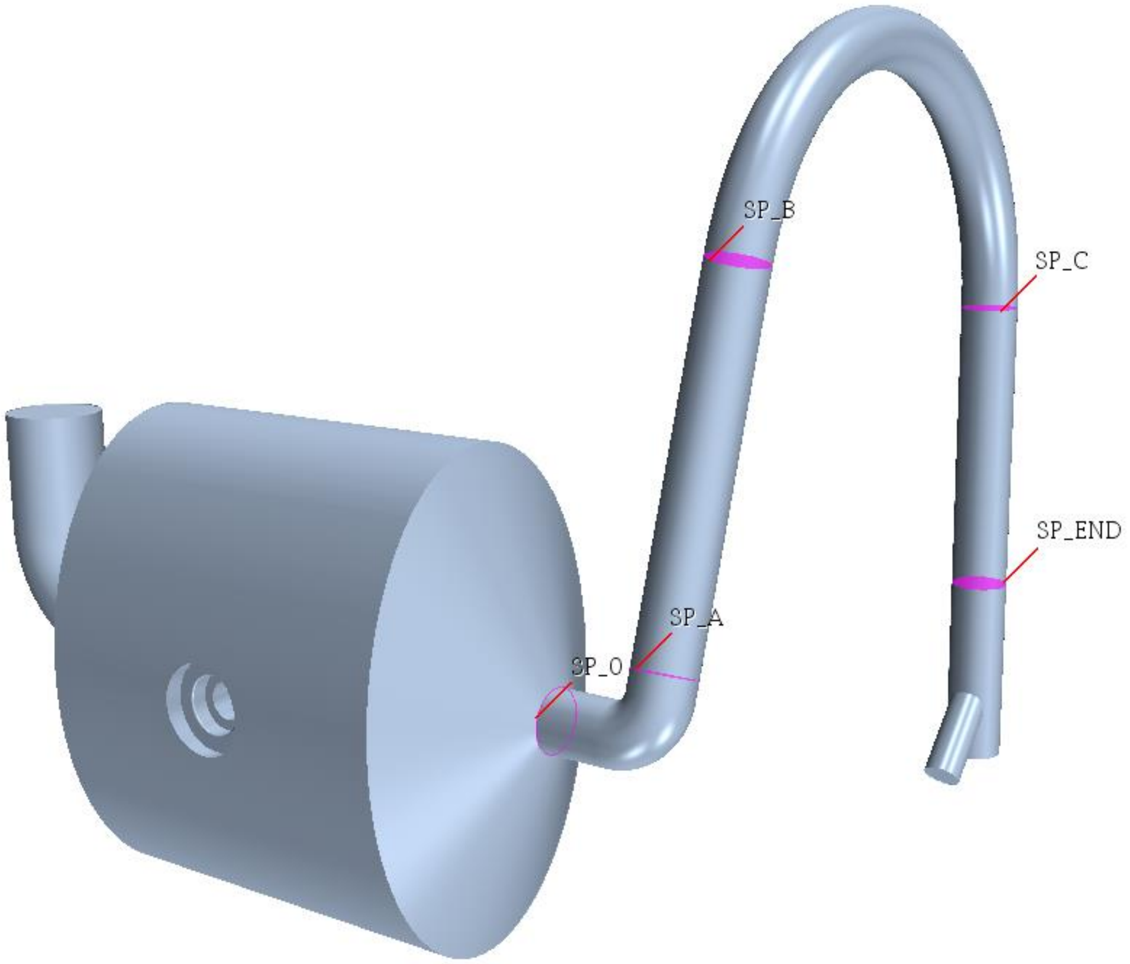
Two main area of particle losses can be highlighted:

A) In the beginning-upper part of the chamber
B) In the low velocity sections of the outlet pipe

Model Assessment

Once the optimized configuration has been chosen, the number of losses are evaluated by measuring the amount of particles at each sampling point of the dyno bench.

Sampling section	**Amount of particles lost [% of total losses]**
SP_0	88.2%
SP_A	0.2%
SP_B	4.9%
SP_C	5.0%
SP_END	1.7%



Results are relative to the case with 1 million of particles injected with a diameter of 1 µm.

Conclusions

A new dust dyno bench has been developed with the support of a 2-phase flow simulation:

- Three main criteria have been considered: design space, particle losses, and isokinetic sampling
- Some design options have been investigated
- Losses area have been identified

Suggestion

- A common CFD simulation methodology should be agreed;
- Real losses should be addressed with experiments (we have measured EF comparable to that found on the most updated literature).

Fabrizio_Venanzoni@brembo.it
+39 035 605 5035
Gabriele_Riva@brembo.it
+39 345 605 5780
+39 035 605 2760

Advanced R&D