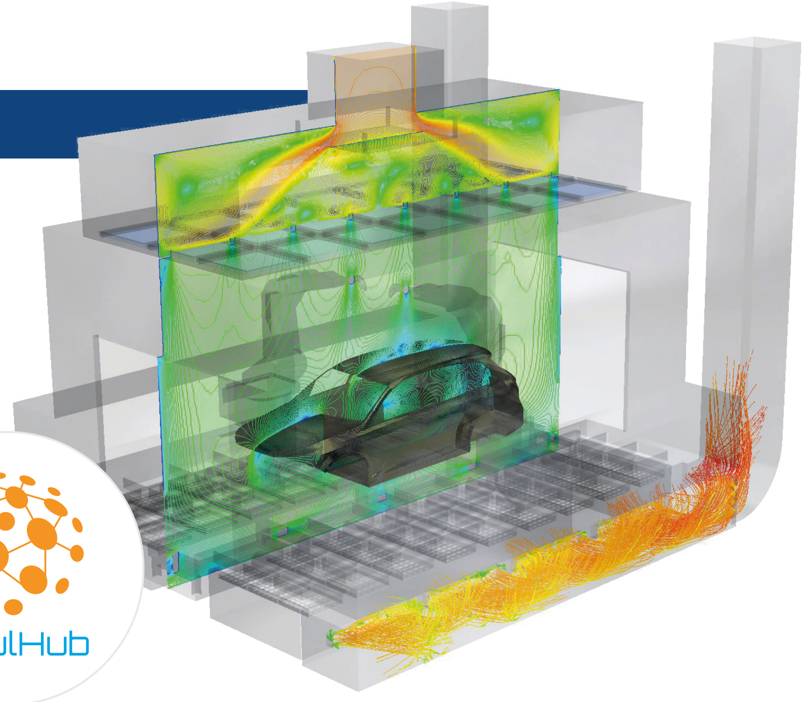


Using CFD simulation to optimize the ventilation system of a spray booth

by Luca Zanellato
SimulHub

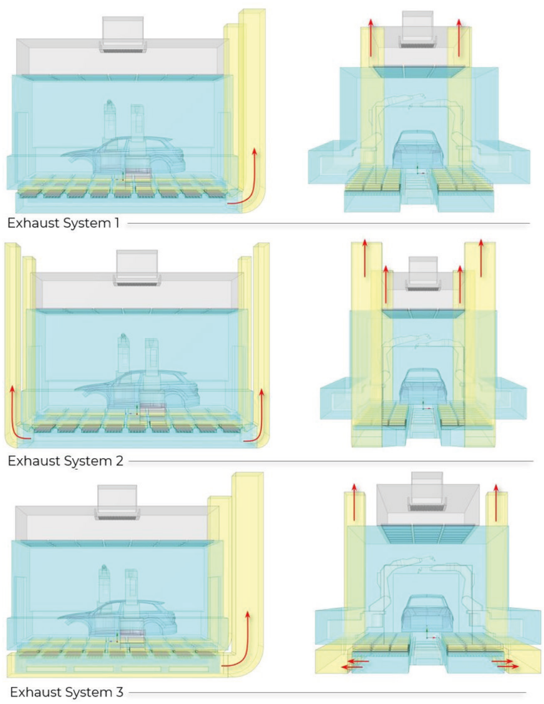
Overspray is a key problem in the paint application process. This phenomenon is caused by inefficiencies of the robots inside the spray booth and can best be solved by balancing the air flow properly in the paint application area.

Ventilation airflow serves two purposes:

- It removes overspray using sufficient down draft;
- It forces the air through a filtration system (known as a scrubber) that separates the paint droplets from the air.

This paper discusses and compares three spray booths which use different exhaust systems. The main objectives of this analysis are to:

- Optimize the exhaust ducts to achieve an even distribution of the air flow while respecting the customer's space constraints;
- Calculate the pressure drop of each design.



1) Two exhaust ducts connect directly to the base of the scrubber.

2) Four exhaust ducts connect directly to the base of the scrubber.

3) Two exhaust ducts connect to two additional ducts located under the robot.

Fig. 2. Three different exhaust system scenarios.

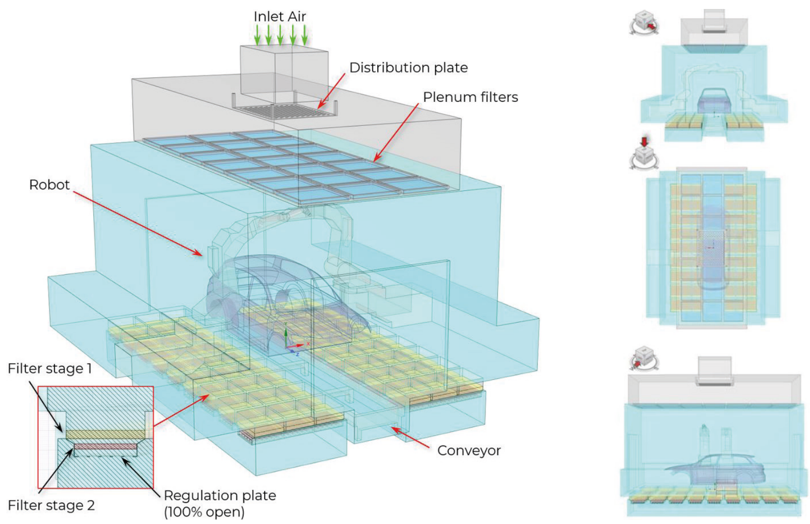


Fig. 1. Geometry of the spray booth.

The simulation was performed with Ansys CFX, a general-purpose CFD (computational fluid dynamics) software. CFD simulation offers the benefit of predicting the function of the proposed ventilation system before it is produced and installed, thus avoiding post-installation modifications and costs to resolve problems.

System description

There are numerous components in the spray booth (Fig.1):

- Ventilation intake duct;
- Air distribution plenum with filters;

- Spray booth including robots, car body and conveyor;
- Scrubber system with two filtration stages;
- Exhaust ducts in three different configurations as shown in Fig.2.

The simulation geometry is comprised of the air volumes while the solid parts are described as boundary walls. Irrelevant details are simplified in order to focus the simulation on the distribution of the air flow and on the pressure losses.

The fluid is air, modelled as a gas with constant properties (at 25°C) and turbulence is modelled using the SST (Shear Stress Transport) model.

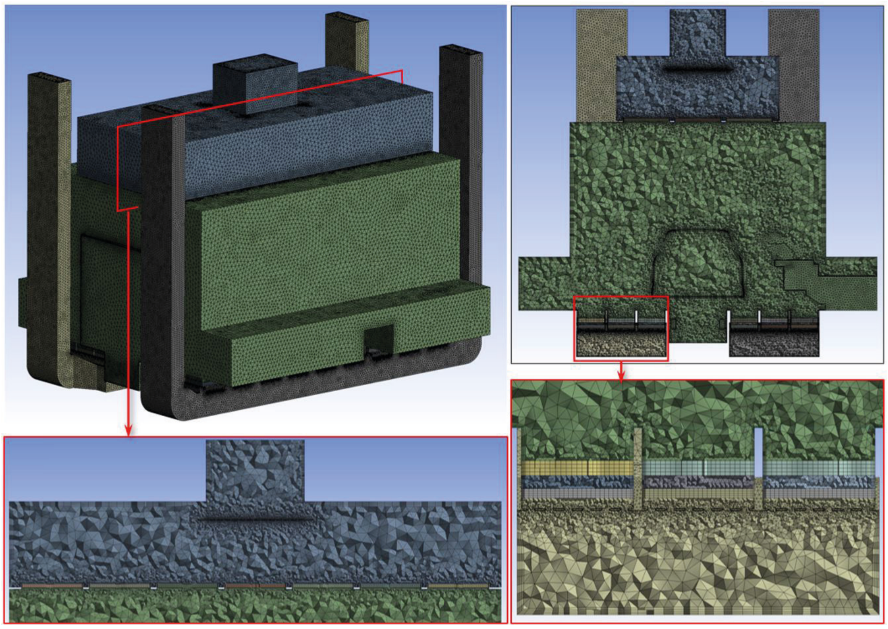


Fig. 4. Mesh composed of 28 million elements.

Name	Boundary	Q	V	P
-	-	m³/h	m/s	atm
Intake duct	Intake velocity	55,500	8.03	-
Outlet duct scenario 1	Outlet velocity	55,500	8.76	-
Outlet duct scenario 2	Outlet velocity	55,500	8.76	-
Outlet duct scenario 3	Outlet velocity	55,500	8.56	1
Booth access gate	Opening pressure	-	-	1

Table 1. Boundary conditions.

Name	DP	Thickness	Q	A	V	K linear
	Pa	mm	m³/h	m²	m/s	kg/m³/s
Plenum	100	50	2,643	1.00	0.73	2,750
Filter stage 1	27	80	1,028	0.46	0.63	550
Filter stage 2	25	50	1,028	0.36	0.79	630

Table 2. Setup of porous domains

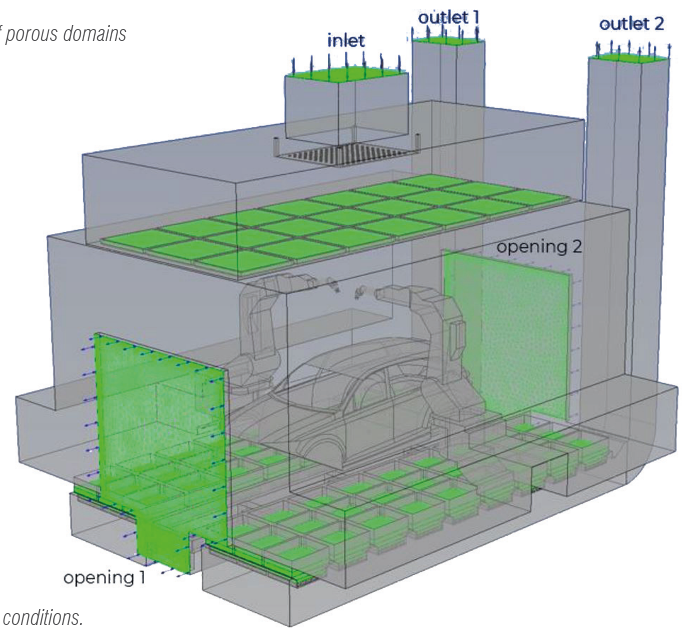


Fig. 3. Boundary conditions.

The filters are modelled as porous domains using porosity and the linear pressure loss coefficient. The boundary conditions are reported in Table 1 and Table 2.

Thereafter, the simulation volume was divided into computational cells using the dedicated mesh-generation tools from the Ansys simulation suite (Fig.4).

Results

To evaluate the downdraft inside the paint application area, the flow rate needs to be checked at the following different heights inside the booth:

- Near the plenum at the top of the booth, air flow must be uniform;
- At a medium height in the booth the flow should accelerate due to the presence of the car body;
- At floor level the air flow over the filters should be evenly distributed.

Scenario 1

The results show that the positioning of the exhaust ducts has a strong influence on the flow distribution inside the booth. The location of the extraction ducts obviously affects the flow distribution over the filters.

Even though this configuration is the most cost effective, the flow is not uniform, and some filters are forced to process more air than their maximum operational limit.

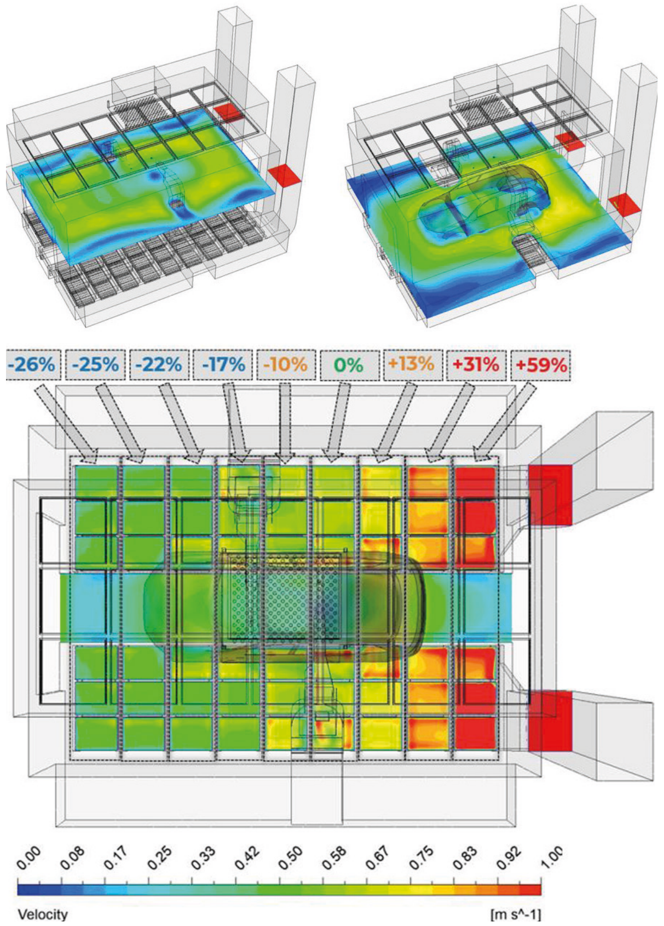


Fig. 5. Velocity distribution for scenario 1.

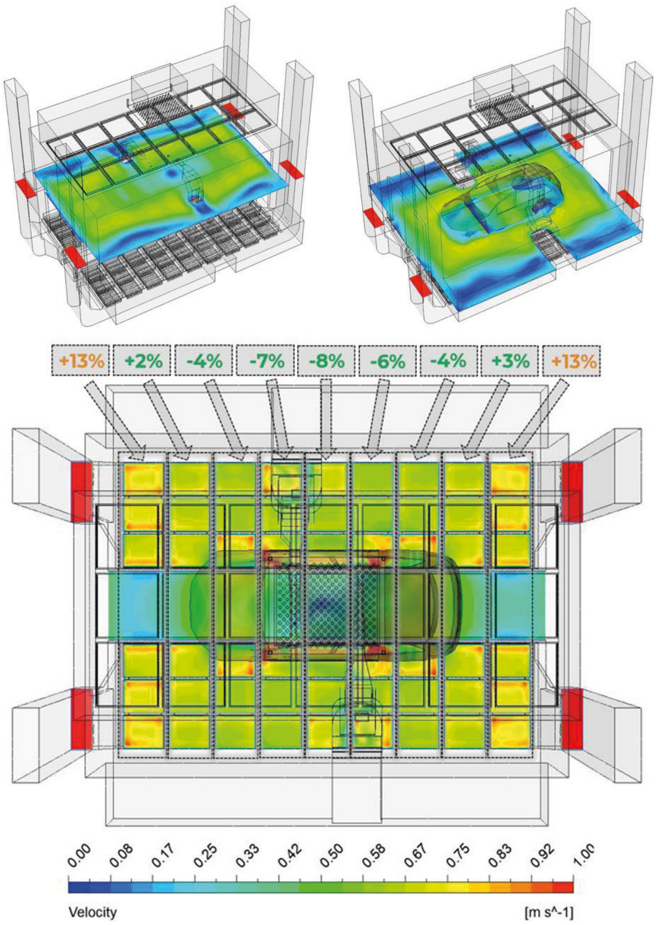


Fig. 6. Velocity distribution for scenario 2.

In these conditions, filters near the exhaust ducts will have a higher load and will consequently be less efficient and become dirty much faster than all other filters.

Scenario 2

As expected, the second scenario has better ventilation. The symmetrical duct design improves the uniformity of the downdraft. The sole concern is that the filters near the exhaust ducts in this scenario are still subjected to a higher load..

Scenario 3

The addition of a second duct below the robots offers the best results, providing optimal air flow distribution. However, this design solution comes at a higher cost (more ducts to install).

The figure shows optimal flow distribution with increased flow under the car body where maximum overspray occurs, thus reducing the risk of the overspray spreading inside the spray booth.

Pressure loss

The following figure graphically represents the pressure losses. They demonstrate that most of the losses are caused by the plenum and the extra loss caused by the additional duct in Scenario 3's extraction system has no significant effect, therefore making the last solution the preferred one from a functional point of view.

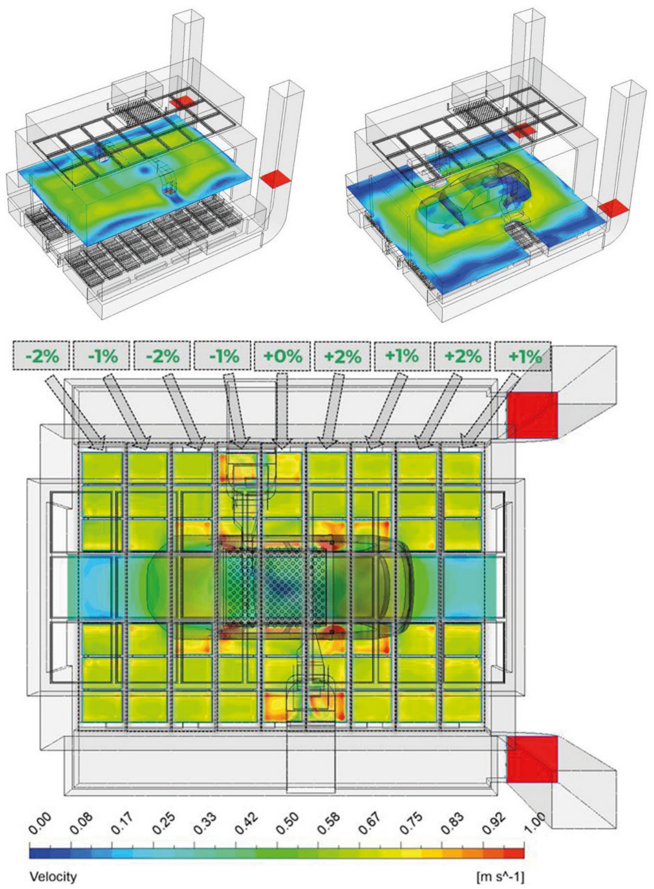


Fig. 7. Velocity distribution for scenario 3.

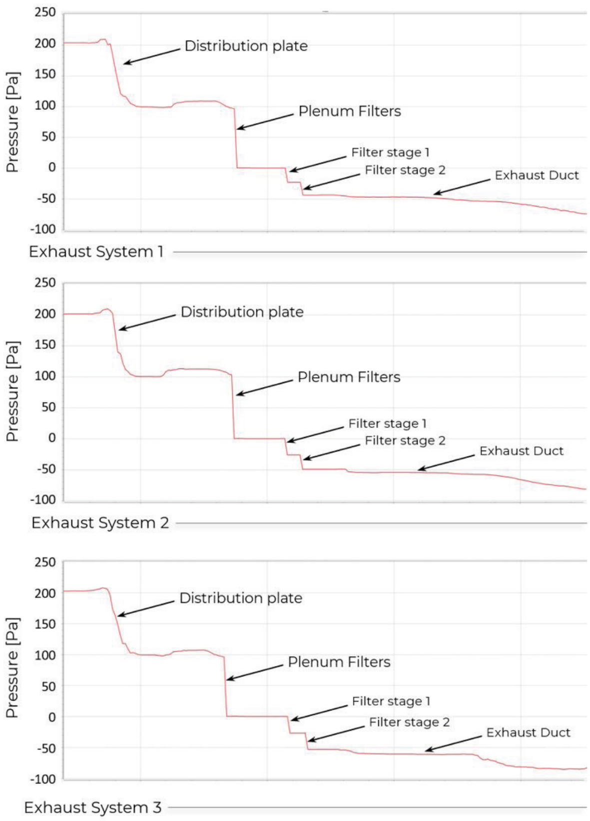


Fig. 8. Pressure loss graphs.

Conclusions

In this paper, we analyzed three different configuration scenarios for the ventilation system of a spray booth.

All three-show similar pressure drops; however, the first scenario has a non-uniform downdraft caused by the position of the extraction ducts. The second scenario has a better airflow distribution during extraction but costs more due to the addition of two further extraction ducts. The final scenario is the best one with the most uniform downdraft.

CFD simulation made it possible to find the best design before the booth was built, thereby improving system efficiency and reducing costs.

About SimulHub

Geico's SimulHub business unit plans, executes, and analyses engineering simulations using professional software in order to predict, prevent, and optimize the operation of production equipment and products throughout the production process. Thanks to its young and highly experienced team of engineers, SimulHub offers CFD, DES (discrete event) and 1D (one-dimensional) simulations accompanied by the valuable expertise gained in the highly competitive and innovative automotive market. SimulHub places its customers at the centre of its existence, adopting their needs as its own and creating tailor-made, sustainable solutions to meet these needs.

For more information:
Luca Zanellato - Simulation Expert
info@simulhub.com