Improving the efficiency of an electric arc furnace’s canopy hood

By Fabio Villa
EnginSoft

This technical article describes a numerical (transient computational fluid dynamics) simulation applied to study the suction efficiency of a canopy hood in a steel plant's electric arc furnace with a view to increasing it. A base case was simulated first after which various geometrical and event modifications were simulated in an optimization loop to identify the best potential geometry to increase the capture of dust from the environment. A standard post-processing procedure was created to easily compare the different cases. The CFD approach was shown to be highly relevant to shorten time to market and reduce the amount of solution testing required.

Using dynamic mesh to evaluate dust distribution

The aim of this study was the quantitative and qualitative characterization of an existing canopy hood configuration for use in a steel plant. The dust extraction in the current device did not seem optimal. Possible improvements that targeted a dust capture efficiency of 90% were studied using the Ansys Fluent computational fluid dynamics (CFD) code.

Mesh model

To start, the geometry was simplified in SpaceClaim to reduce the size of the mesh. The resulting geometry (Fig. 1) was used to realize the base case (the standard case).

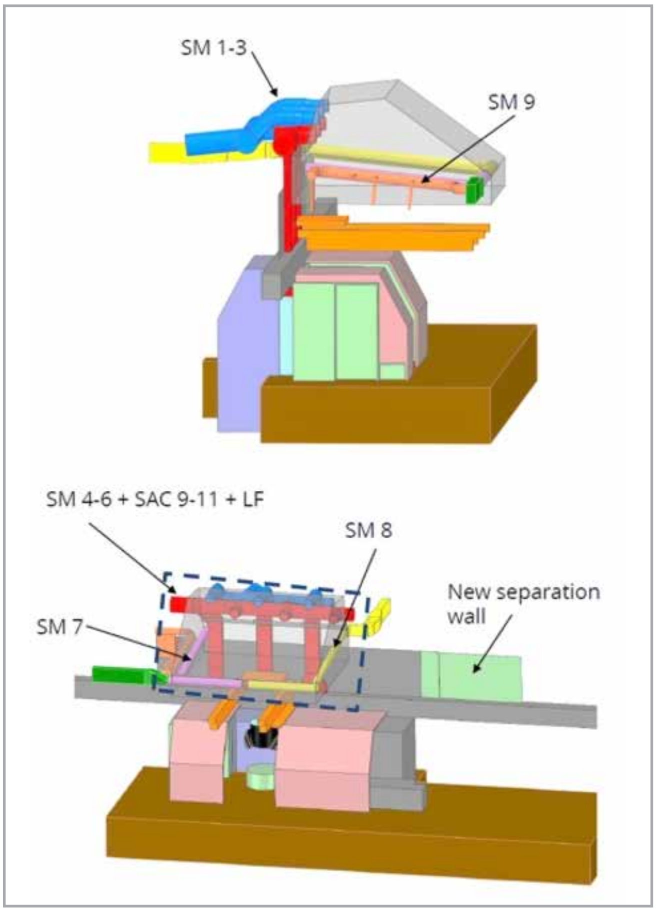

Fig. 1 – Base case geometry after simplification

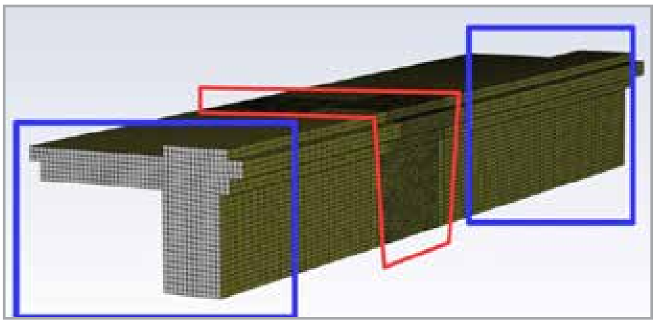

Fig. 2 – Box with hexahedral mesh for the moving parts

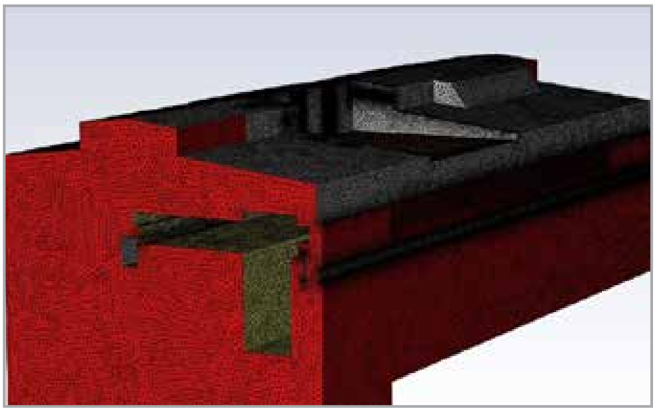

Fig. 3 – Static tetra mesh

The geometry included the canopy hood and all the tube connections. Each connection was named to evaluate the relative suction efficiency.
The mesh consisted of a hybrid mesh (tetrahedral and hexahedral elements) of about 13 million elements. The mesh was divided into moving and static parts.

The layering method for dynamic meshes was used since only the translation movement had to be performed. For each moving part, the following procedure was developed during meshing:

- Each movable solid part was included in a box mesh (red in Fig. 2) with tetra elements. A sweep mesh was used along the direction of translation (blue in Fig. 2) to apply the layering method.
- The static mesh was created with tetra elements (Fig. 3).
- These two cell zones were combined and imported into Fluent to complete the computational domain. Mesh interfaces were created to connect the cell zones.

The CFD model

Ansys Fluent 19.0 was used as the solver for this study. A transient solution was required with a total simulation time of 130 seconds. The high temperature range in the computational domain made it necessary to activate the energy equation and resolve the temperature fields.

The layering method was used for the dynamic mesh zone. This method is recommended for translation (or a combination of translation movement). The layering approach ensures a faster dynamic mesh compared to other techniques. For this case, the position coordinate (x,y,z) profiles for each moving part were defined previously (t=0-130 s).
The complete process consisted of three steps:

- In step 1 (Fig. 4), the doghouse opens and the crane carries the basket into the oven.
- In step 2 (Fig. 5), scrap charging is activated. The geometry of the basket is changed (the basket is empty in this step, so the flow can pass through it. The dust begins to fill the environment and part of this dust is captured by suction. The iso-surfaces of the particle mass concentration can be used to display the dust cloud. Finally, the crane carries the basket outside the furnace again.
- In step 3 (Fig. 6), the doghouse closes and the crane carries the empty basket away. In this step, most of the dust is finally extracted and disappears from the domain. However, some dust can escape from the canopy hood. This study aims to improve the dust capture by suction.

The transition between the steps required a change in the direction of linear motion. Therefore, three meshes were created. In each mesh, the components were set in the initial position of each relative step. The main mesh parameters were retained across all three meshes. The sweep mesh was clearly modified to match

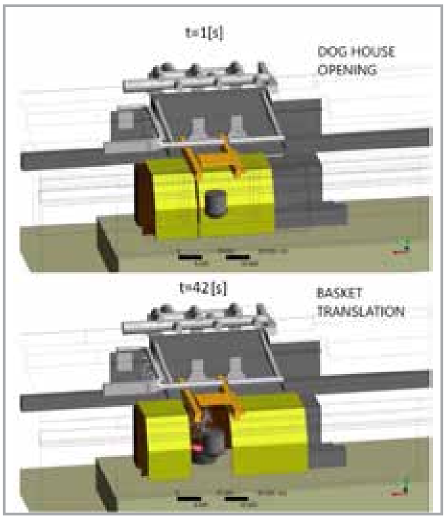

Fig. 4 – Step 1

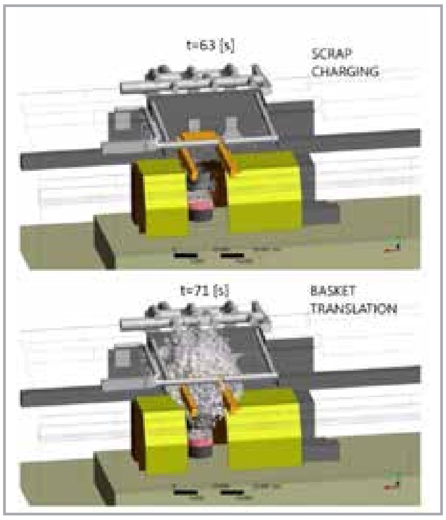

Fig. 5 – Step 2

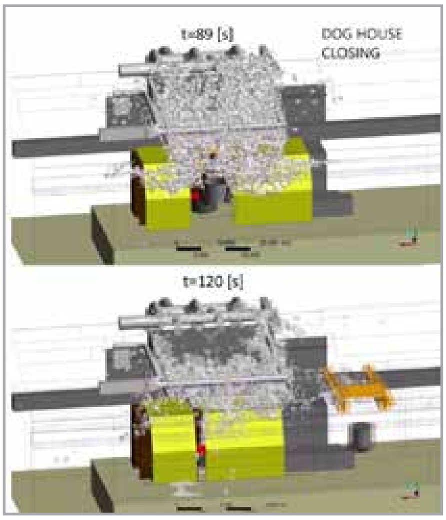

Fig. 6 – Step 3

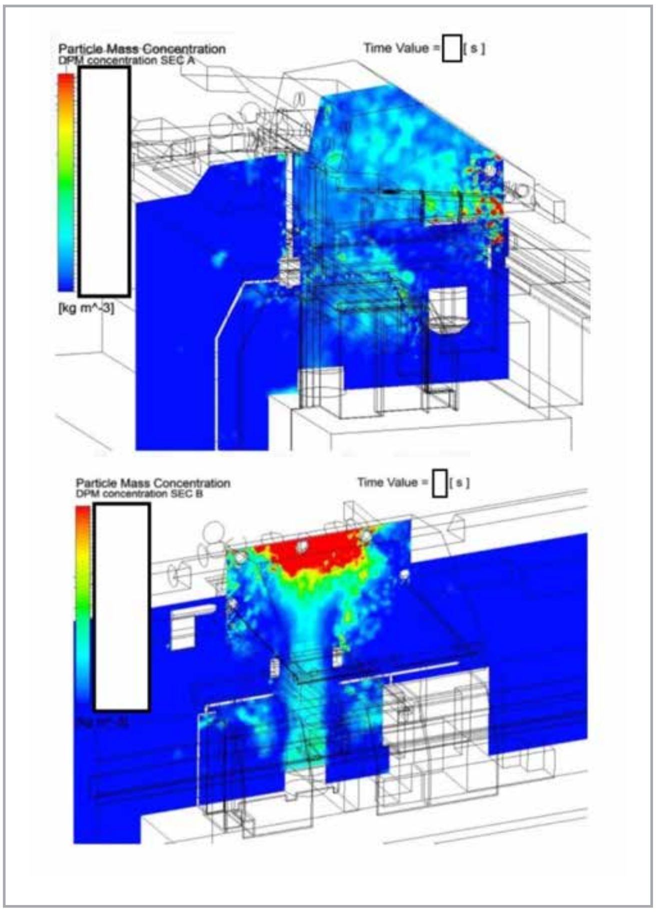

Figure 7: Particle mass concentration

the correct direction of motion. The Events module of Fluent's dynamic mesh models was used to configure the events according to the current step. This procedure allows you to use the same setup for all the transient simulations. The transition between the steps required a mesh change: it was performed manually, but could also be performed using an appropriate journal file in Fluent. The dust was modeled using the discrete phase model (DPM) approach. DPM is a Lagrangian framework to simulate discrete particles (dust) in a continuous fluid domain (air-gas). The furnace is the source of the dust. The Weibull Size Distribution of the dust was defined by the composition of the scrap. A velocity and total flow rate profile was used. The inertial particle model could take into account the thermal effect of the discrete phase. The canopy hood comprises four mass-flow outlets, with different activation times and mass-flow values. The suction sequence was controlled by prescribed profiles. A pressure outlet (atmospheric pressure condition) was applied to the surfaces adjacent to the external environment.

Several report definitions were created in the domain to evaluate the split of the mass flow in the suction system. This helped to identify potential areas for improvement.

Base case results

Post-processing was used to evaluate the efficiency of the current geometry. The following data was saved during the execution of the transient case:

- Report definitions to track the average condition of inlets/outlets over time. Tracking variables included temperature, injected mass of dust and the escaped mass of dust.
- CDAT files were stored every five seconds to obtain the most important fields (velocity, particle mass concentration) as a function of space.
- Particle history data file: a data file, available in Ansys Fluent, which includes all the most important particle information (can be import directly in Ansys CFD-POST).

A significant variable for this study was the Particle Mass Concentration (PMC). It represented the mass concentration. Figure 7 shows the particle mass concentration in two plane and in two different simulation time. Suction of the canopy hood is clearly visible.

After identification of the PMC evolution in the domain, check of the velocity field in several sections could highlight possible critical zones (e.g. with a low suction efficiency) and suggest some geometrical changes.

In Figure 8 velocity magnitude was plotted in two different planes, in order to visualize the flow velocity inside the channels.

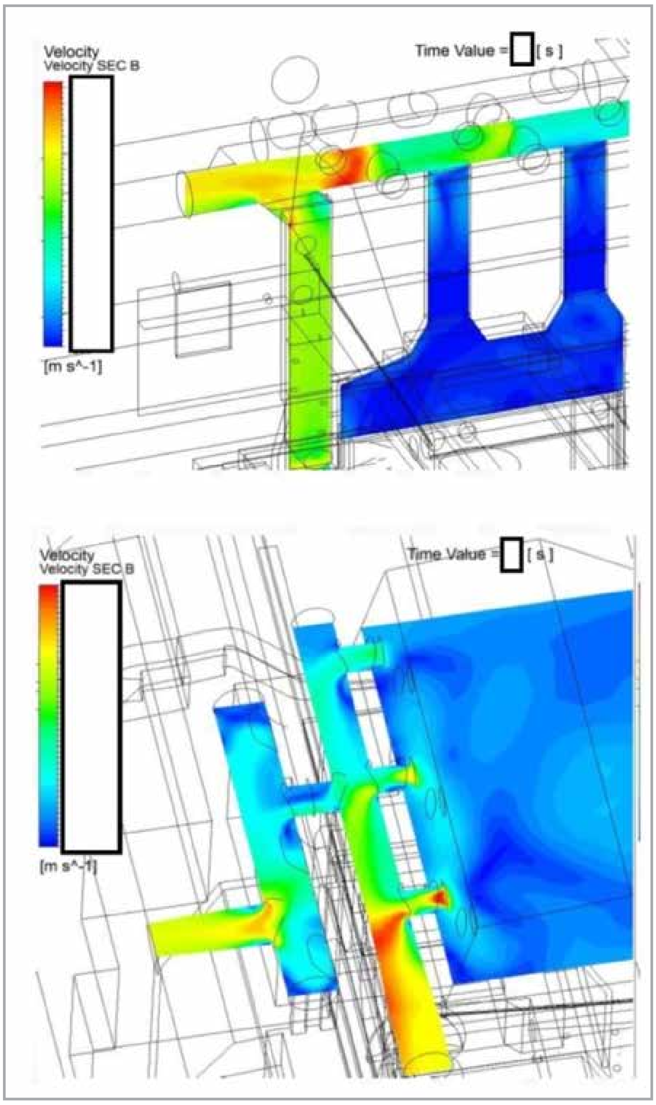

Figure 8: Velocity profile

About the Redecam Group

The Redecam Group is an Italian company specialized in the design, manufacture and installation of industrial plants for environmental protection, ranging from simple filtration equipment to complex flue gas treatment systems.
Redecam's team of highly-qualified engineers offers tailor made turn-key air filtration and flue gas treatment (FGT) solutions, helping customers worldwide to achieve their reduction targets for air emissions rapidly and cost-effectively. In 2020, the company is celebrating 40 years since its establishment during which time it has built a strong track record of more than 2,800 references in almost 100 countries and on every continent, including Antarctica.
Redecam's business covers a broad spectrum of industrial sectors, including cement, lime, waste-to-energy (WtE), and biomass. The company has a dedicated portfolio of technologies for the metal industry and is proud to help this important sector to comply with the most stringent environmental regulations.

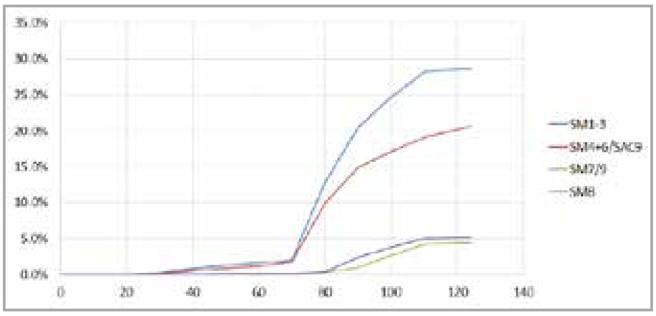

Figure 9: dust% cumulative

Finally the efficiency of the current geometry can be evaluated using the dust% cumulative. This percent variable represents the cumulative value of the escaped mass (during time) compared to the total injected mass. It is possible to see in Figure 9 the chart for this variable, evaluated in the four outlets, during the complete simulation.
After a rapid increasing of the suction effect, from about 70 to 110 seconds, the cumulative function reached a plateau value, always less than 100%. Main goal of the optimization process was to increase this plateau maximum value (e.g. increase the suction efficiency).

Optimization results

Optimization process, as applied in this case, can be summarized in the following optimization circle:

1. Check of the results of the previous case. As describe in Base Case Results paragraph, the post process helps to identify the possible geometrical/event modifications.
2. Implementation of the identified geometrical change. Depending on the specific change, some revision of the mesh configuration or cell zone/boundary condition names may be required. However, the main settings remain identical and can be reused.

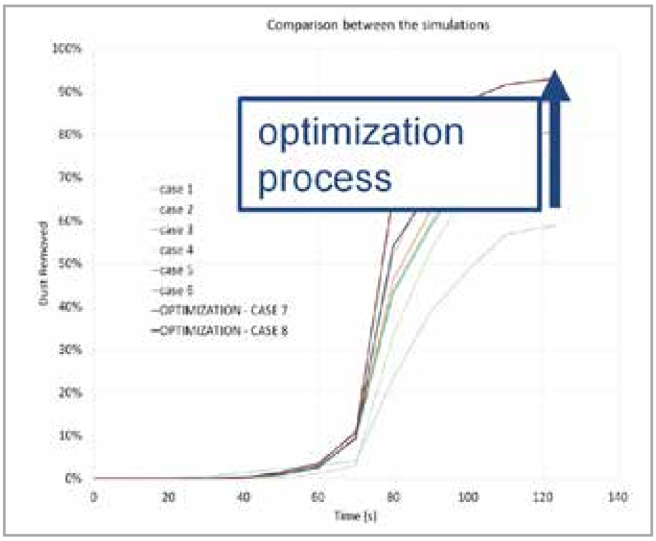

Fig. 10 – Optimization process

3. The new Fluent case is effectively updated and executed.
4. Post-processing of the new case is performed. The graphs of particle mass concentration, velocity, and cumulative percentage of dust for the new and previous cases are compared.

This loop was repeated several times, also based on the technician's experience. Ultimately, the best result, as shown in Fig. 10 (total dust capture of 93%, compared to about 60% in the base case), confirmed the possibility of increasing suction efficiency by using well-defined geometrical/event modifications.

Conclusion

This article presents a numerical model for optimizing the suction capacity of a canopy hood used in a steel plant. The transient CFD simulation was designed using Ansys Fluent. The numerical model included a dynamic mesh of different cell zones where an appropriate meshing technique made use of the layering method for dynamic mesh and of the Discrete Phase Model to track the dust clouds.

A standard post-processing procedure was created to easily compare different cases. After simulating the existing situation (called the base case), some geometrical/event modifications were applied and tested in a optimization loop. The best configuration obtained showed a total dust capture of 93%, compared to about 60% in the base case. The possibility of increasing suction efficiency with well-defined geometrical/event modifications using the CFD approach, as conducted in this study, appears to be highly relevant to shorten time to market and reduce the amount of solution testing required.

For more information:
Fabio Villa - EnginSoft
f.villa@enginsoft.com