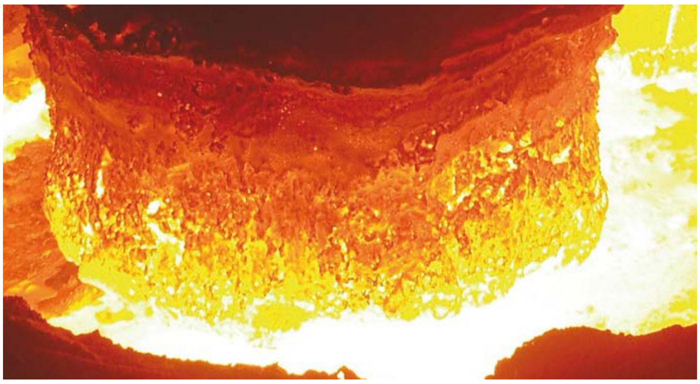
Case Study

SMS Mevac UK Ltd

CFD Analysis of a Fixed Fume Extraction Hood

Company Profile

The term "Mevac" in the company name comes from two sources: Standard Messo, founded in 1960, and Vacmetal, founded in 1969, both providing pioneering solutions for secondary metallurgical processes. They merged to form Mevac in 1999.



SMS Mevac deliver highly efficient, innovative technologies and services for the refining of liquid steel. To achieve this goal, they balance process technologies, market requirements, customer demands and environmental responsibilities. This includes engineering, equipment supply, supervision of erection and commissioning, as well as training and after sales services.

SMS Mevac have a world-wide leading market position based upon their unique metallurgical and process expertise.

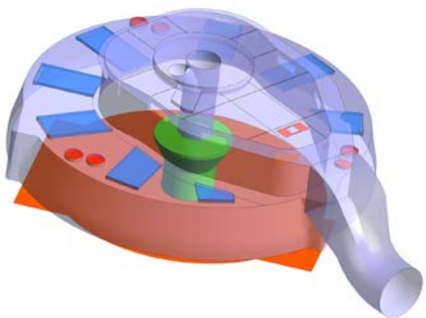
Background

A fume hood is a local ventilation device designed to limit exposure to hazardous or noxious fumes, vapours or dusts. Two main types exist, ducted and recirculating. The principle is the same for both types: air is drawn in from the open side, and either expelled outside the building or made safe through filtration and fed back into the workspace.

Most fume hoods for industrial purposes are ducted. Hazardous or noxious vapours, gases or dusts are removed from the workspace and dispersed into the atmosphere.

The correct flow of air from the workspace into the hood is critical to its function. Much of fume hood design and operation is focused on maximising the correct containment of the air and fumes within the fume hood.

Analysis



SMS Mevac UK Limited required a Computational Fluid Dynamics (CFD) analysis of a Fixed Fume Extraction Hood in order to understand the flow inside the hood, with a view to optimising its design.

Two CAD models of the fume extractor hood were supplied by SMS Mevac. The models had different sized slots in the horizontal and upper sections. The geometry was defeatured appropriately for CFD analysis and a fluid domain was derived for the inner portions of the hood. Some of the external/internal features of the hood which would not affect the flow were removed in order to mesh the model.

A boundary layer with an appropriate thickness was employed for the fluid domain walls to help capture the flow variations near the wall. In addition, a finer resolution mesh was placed on surfaces where there was a rapid change in fluid parameters or the geometric dimensions were relatively small.

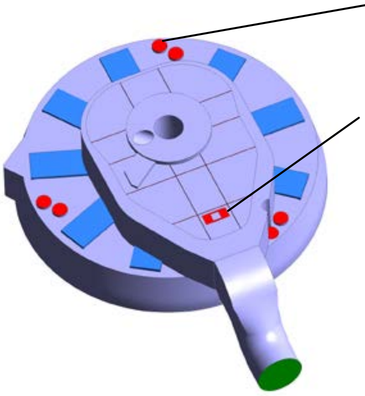
A steady state, single fluid, non-buoyant, incompressible and isothermal CFD simulation was carried out in ANSYS CFX.

Boundary Conditions

The boundary conditions applied to the two models were as follows:

- The inlet was specified as an 'opening' boundary condition which allowed the fluid (in this case, air) to enter as well as leave the domain.

- A volumetric suction flow rate of 50,000m³/h was applied at the main outlet of the hood.
- Wall boundary conditions were given to all other surfaces through which there is no flow.



These are pipes where wire ropes pass through. The ropes were not modelled because of their very small sectional area (compared to the dimensions of the pipes) and effects to the flow were thought to be negligible. Due to the different pressure between the interior of the fume hood and the outer environment, some flow was expected through the pipes. For this reason an Opening Condition was set at the holes on the top surface.

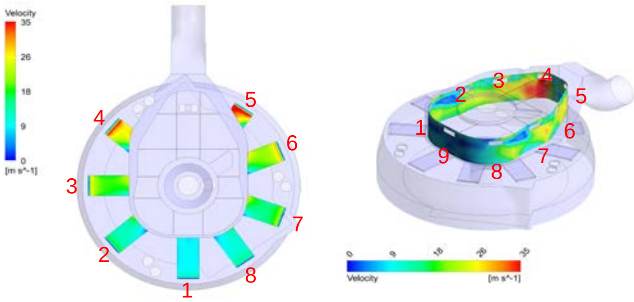
Hoses and cables pass through this hole. To simplify the model, a small square surface was modelled in the middle of the hole, in order to reduce the free area through which some fluid can flow. An Opening Condition was set.

The suction of the fluid from the bottom of the fume hood was obtained setting a mass flow rate on the outflow of the duct.

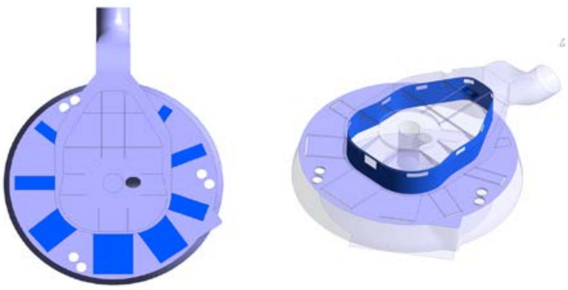
The fluid was modelled as air at standard conditions of $\rho_{air} = 1.185 \frac{kg}{m^3}, p = 101325 Pa, T = 25^oC$

The flow rate through the inner pipe and the conical hopper was not modelled. These were included as obstacles to the incoming fluid from below. An opening condition was also set around the outside of the ladle cover, in order to allow 'external' fluid to be sucked inside the hood.

Original Design



Modified Design

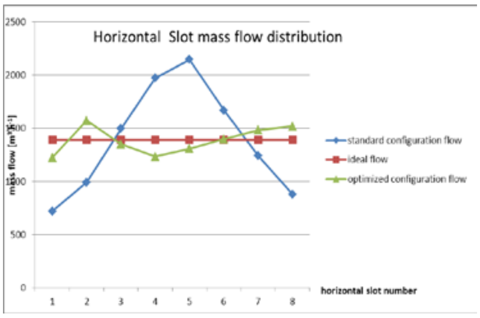


It was noted that the velocity of flow was higher around the horizontal slots closest to the exhaust duct, and the same was found to be true of the vertical slots – where flow velocity was seen to be higher around the exhaust duct also.

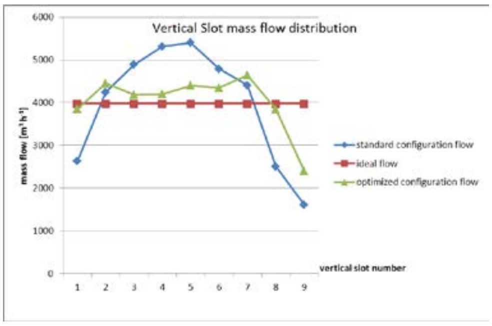
Based on the above results from the original design, the geometry of the slots was modified in order to try and gain a more balanced flow throughout the hood.

The graphs below show the flow through each slot for both the horizontal (below left) and vertical slots (below right), for both the original and optimised configurations. It can be seen quite clearly that the optimised configuration showed an improvement in flow distribution.

Horizontal Slots

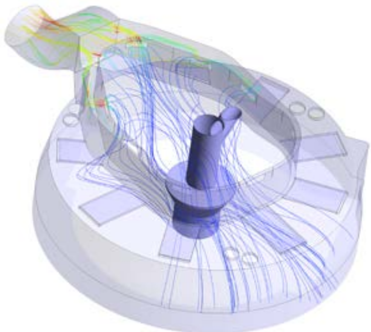


Vertical Slots



Design Benefit

Due to the relative distances between the slots (both horizontal and vertical) and the exhaust duct, it was noticed that there was a non-uniform distribution of the flow through these slots. Using the results obtained from the original design of the fume hood, the area of both the vertical and horizontal slots were modified and a more uniform mass flow distribution was obtained. Apart from the horizontal slot no.1 and the vertical slot no.9, all the other slots responded to the variation of their sectional area. The different behaviour of the horizontal slot no.1 and the vertical slot no.9 was attributed to their position relative to the exhaust duct.



IDAC, Airport House, Purley Way, Croydon, Surrey, CR0 0XZ, England
Tel: +44 (0)844 212 5900 **Fax:** +44 (0)844 212 5910
Email: info@idac.co.uk **Homepage:** www.idac.co.uk