Exhaust hood study – Test bench project for exhaust hoods' energy characterization

Zita Dias Carreira

Department of Mechanical Engineering, Instituto Superior Técnico, Universidade de Lisboa

ABSTRACT

The present work consisted in characterize exhaust hoods, being developed a bench test to determine performance curves, working points and fluid dynamic efficiency. Installation setup was designed according to international standard IEC 61591 and consisted in a reference exhaustion pipe, a compensation chamber, a flow measuring system and an auxiliary fan. The characteristic curves were determined by plotting the pressure for different volumetric flow rates, which were obtained by applying different pressure losses. Pressure losses were controlled according to the selected orifice plate, installed mechanical valve position and frequency supplied to the auxiliary fan. The operational range was validated through the evaluation of the installation line pressure drops and the exiting velocity profile. In order to understand the compensation chamber influence on the bench test flow, a laser visualization technique was applied on the outlet exhaust hood flow. A computational fluid dynamic tool was used to estimate uncertainties associated with the numerical modulation for different internal flow regimes. The particular case of an exhaust hood inner flow was simulated and a qualitative influence between its geometry and the velocity profile was inferred.

Key-words: Exhaust hood, performance curves, fluid dynamic efficiency, IEC 61591, code verification, solution verification.

1. INTRODUCTION

Exhaust ventilation systems usage in kitchens and other spaces is common practice nowadays. These types of local ventilation systems are prepared to remove mass and thermal load produced by any pollution source before they spread. There are three types of ventilation systems: namely mechanical, hybrid or natural [1]. A mechanical ventilation system is the one being addressed, incorporating one or more fans, which are responsible for the air flow.

In a simplistic way, it can be assumed that the global efficiency of the device depends only on the ratio of the amount of contaminant captured by the hood to the total amount produced by the source. The capture capacity of contaminants is directly related to the capture capacity of the exhaust hood, and, consequently, to its extraction flow, Q. This flow depends on exhaust hood's geometry, fan's configuration (such as the number of blades and its angle), and fan's location inside the hood. According to the international energy savings tendencies, in 2015 a new legislation comes into force, requiring that every exhaust hood on the market has an energetic label, containing information about its energetic class, fluid dynamic efficiency, among others. To determine the fluid dynamic efficiency, exhaust hoods' performance curves are required, which can be obtained through experiments in a specific bench test. In the present work are presented the chosen bench test components, as well as the procedure to draw performance curves and calculate the fluid dynamic efficiency curve, being applied to a practical case, a
TEKA TL1-62 exhaust hood model. To understand hood’s fluid flow dynamic, a laser technique is used. A numerical analysis of the uncertainties associated with internal flow regime simulations is presented, with the example for the same exhaust hood model, TL1-62.

2. EXPERIMENTAL SETUP FOR EXHAUST HOOD CHARACTERIZATION

2.1. Exhaust hood model

TEKA TL1-62 model is composed by two independent grease filters and an interior radial fan containing fifty backward curved blades, located tangentially to the hood’s exit. The exhaust hood has two working positions, corresponding to two different fan’s angular velocities, corresponding to two different catalogued airflows, which are 233 m³/h and 332 m³/h.

2.2. Experimental setup

In order to perform exhaust hood’s characterization, it was designed and built an universal bench test, according to IEC 61591 [2], illustrated in Figure 1, consisting of the following components:

1. The exhaust hood in analysis with an exit diameter of 120 mm;
2. An inox exhaust pipe, whose length must be five times exhaust hood’s exit diameter; In the present case, 600 mm long with an internal diameter of 120 mm, which allow static pressure measurements along its length.
3. A compensation chamber intended to eliminate the flow’s disturbances, with an internal diameter of 135.6 mm;
4. A transition cone, which reduces the diameter from 136 mm to 69 mm;
5. A PVC pipe, about 813.6 mm long, with an internal diameter of 69 mm;
6. A connection flange, with a flow measurement system in between. Upstream fully developed flow condition is a requirement;
7. A PVC pipe, 1017 mm long, with an internal diameter of 90 mm;
8. A mechanical valve to control the airflow;
9. An air blower (Busch, mod. SAMOS SB530D, 4 kW power) to compensate all the pressure losses caused by upstream installation elements, such as the pressure compensation chamber and the device used to measure the flow.

Besides the hood’s model tested, the bench test is prepared to characterize other hoods models. Downstream from the exhaust tube, there is a compensation chamber responsible for removing flow disturbances. The compensation chamber velocity depends on its size and composition, being the ideal the one whose inner velocity is minimal enough, sufficient to dissipate all flow disturbances.

Figure 1- Experimental facility diagram for the study of exhaust hood performance: 1=Exhaust hood, 2=Exhaust pipe, 3=Pressure compensation chamber, 4=Transition cone, 5=PVC pipe, 6= Connection flange, 7=PVC pipe, 8= Mechanical valve, 9= Air blower
Several tests were performed with different pressure compensation chamber configurations and it was concluded that the most efficient way to obtain a fully developed profile in (6) is a camera with a diameter of 135.6 mm and a volume of 5776 mm$^3$ filled with small spheres that, conceptually, represent a porous medium. However, despite being the most efficient arrangement to obtain a fully developed flow, it has the disadvantage of introducing a higher-pressure drop. Downstream connection of the compensation chamber consists in a stainless steel cone, which causes flow acceleration that must be compensated by an extra length, downstream, so fully developed flow is obtained before flow rate measurement section. As a reference, it was used a typical length tabulated in ISO 5167-2 [3] for a reducing component diameter, from 2D to D in the present case.

An orifice plate was used for flow measurements, characterized by a given $\beta$, which is the ratio of pipe diameter to device throat. A $\beta$ of 0.678 was chosen, being remaining dimensions established according to ISO 5167-2 [3]. For differential pressure measurement, essential to calculate the flow, two static pressure tappings were used, located at 69 mm upstream and 35.4 mm downstream from the device location.

2.3. Fluid flow measurement

Fluid flow can be calculated through the area integral

$$Q = \int_s \vec{v} \cdot \vec{n} \, ds$$

where $\vec{v}$ corresponds to the flow velocity and $\vec{n}$ to the normal to the surface.

There are several techniques to measure fluid’s volumetric flow, but for industrial purposes it is common to use differential devices, as they are cheap and the technique to calculate airflow is simple to use. In this work an orifice plate was used, which is, essentially, a thin plate with a centred hole mounted in a pipe.

The results obtained with the orifice plate were compared with the flow calculated through equation (1).

The working principle of these components is based on acceleration of a fluid stream through some form of nozzle, as shown schematically in Figure 2. The flow separation at the sharp edge of the nozzle throat creates a recirculation zone, as shown by the dashed lines downstream. The mainstream flow continues to accelerate from the nozzle throat to form a vena contracta at section 2, and then decelerates to fill the duct.

Figure 2- Differential device example-nozzle

An expression for the optimal flow rate can be obtained, assuming a uniform flow and an incompressible fluid, by applying the mass conservation and energy balance between sections 1 and 2 as

$$Q_{ideal} = A_2 \sqrt{\frac{2(p_1 - p_2)/\rho}{1 - (A_2/A_1)^2}}$$

where index 1 denotes the station upstream the plate and index 2 is at the vena contracta. This equation describes the idealized flow rate.

Considering C as the ratio of the actual flow rate to the ideal flow rate, it can be calculated using the expression proposed on ISO 5167-2 [3]. For compressible flow, including a correction in terms of the expansibility $\varepsilon$ of the fluid, previous equation takes the form

$$Q = C \varepsilon A_2 \sqrt{\frac{2(\Delta p)/\rho}{1 - (\beta^3)}}$$

where $\Delta p$ is the pressure difference across the device.
2.4. Pressure measurement

Dynamic pressures measurements were made with two purposes:
- Measuring the velocity on the PVC exit, upstream to the measuring device, in order to evaluate velocity's profile;
- Validate the system’s flow measurement comparing the flow rate calculated using equation (3) with the flow rate calculated by velocity profile integration from equation (1).

Compensation chamber selection was made mainly through evaluation of the pressure across the installation without the orifice plate, since in a tube with constant diameter, according to the energy equation, the pressure drop between two generic points, 1 and 2, must be linear. By measuring the static pressure along the PVC pipe, if it is found that the pressure drop is effectively linear, the velocity profile is fully developed, which means that the length of the PVC pipe, upstream of the orifice plate, together with the pressure compensation chamber are sufficient to obtain a fully developed flow and validate the pressure measurements.

Experimental setup is equipped with 11 static pressure tappings, as shown schematically in Figure 3.

2.5. Operating points and fluid dynamic efficiency

In this work, hood evaluation is mainly made using pressure versus flow curves. The pressure to be used corresponds to the pressure drop in the exhaust hood and must be registered in pressure tapping number 2. Pressure measured on tappings number 8 and 9 are used to calculate the airflow.

Flow control can be done in two ways, either using the frequency variation, which the vacuum pump is powered by, or by actuating the valve installed in the PVC pipe, as shown in number 8 of Figure 1.

To determine the working points it is necessary to intersect the curves previously obtained with the installation curve, where the hood is installed.

IEC 61591 [2] defines the installation’s curve for which the working points must be calculated. For an exhaust hood outlet diameter of 120 mm, it yields

$$\Delta P \ [Pa] = 0.000375Q^2$$

(4)

Since the exhaust hood in study has two operating velocities two characteristic curves were traced, resulting in two working points when intersected with the installation curve provided by the standard.

It is also possible to draw a graph of exhaust hood's efficiency as a function of the flow rate. The efficiency curve represents the relationship between the energy transferred into the airflow and the power supplied to the motor. As flow rate increases, efficiency increases towards the best efficiency point (BEP). After this peak, with the progressive flow increase, efficiency starts to decrease. The ideal scenario would be working near the BEP flow, in order to avoid losses and consequent device inefficiency.
In Regulation 66-2014 [5] for energy certification, a parameter related to the fluid dynamic efficiency, FDE, is used, which can be calculated by the following equation

\[ FDE = \frac{Q \times P_{\text{stat}}}{3600 \times W} \times 100 \tag{5} \]

2.6. Laser visualization

As mentioned in section 2.2, laser usage was related to the need to visualize the exhaust outlet flow, to better understand the compensation chamber’s pressure effect on flow stabilization. The laser used was a 8 W Spectra-Physics, Series 2000, Model 2170. In the experimental setup a cylindrical glass was used, whose purpose is to open the light beam transforming it into a plan that would intersect the exhaust hood central plane. This setup allows airflow visualization in a plane that was aligned with the hood exit without any exhaust pipe.

3. RESULTS PRESENTATION AND DISCUSSION

3.1. Performance curves

3.1.1. Fluid flow measurements validation

Measuring system validation was performed using the experimental setup depicted in Figure 1 without the blower installed. To validate the flow measurement method is necessary to ensure that the upstream flow at the orifice plate is fully developed.

The velocity profile measured, based on the dynamic pressure, at PVC pipe outlet represented with the number 5 in Figure 1, preceding the measurement device, is represented in Figure 4 for the first fan’s velocity. Results found for the second fan’s velocity were similar.

Both profiles follow a logarithmic law that can be adjusted to a power law, characteristic of turbulent regime, according to equation (6), with \( n \approx 10 \) and \( y = R - r \), where \( R \) is the tube radius and \( U_{\text{max}} \) is the maximum velocity in circular cross section.

\[ \frac{u}{U_{\text{max}}} = \left( \frac{y}{R} \right)^n \tag{6} \]

The evolution of the static pressure along the installation without the vacuum pump and the orifice plate was also registered, being the results presented in Figure 5, with \( R^2 = 0.9706 \) for the first fan velocity and \( R^2 = 0.9515 \) for the second one.

In summary, two previous experiments show that the PVC pipe length together with the action of the pressure chamber are sufficient for the flow which reach the orifice plate to be almost fully developed. For the installation in Figure 1, with differential devices installed, static pressures at each pressure tap were measured and plotted in function of distance to the exhaust hood, as shown in Figure 6.
Two large pressure losses were observed, the biggest one occurring in the compensation chamber and the other in the differential device, used to measure the flow. The largest pressure loss introduced by the pressure stabilizer justifies the need to install a vacuum pump on the bench test in order to compensate that loss.

The differential pressure obtained between upstream and downstream orifice plate tappings was 69.42 Pa for the first exhaust fan velocity and 83.11 Pa for the second exhaust fan velocity. Using equation (3), volumetric flows of 47.48 m$^3$/h and 50.25 m$^3$/h were determined for the exhaust fan velocity 1 and 2, respectively.

Using the installation with the differential device installed, the velocity profile was measured along a length of four radius of outlet PVC pipe, number 7 of Figure 1, and volumetric flows were calculated with the velocities integration method.

Comparing the values obtained, based on the differential pressure on the orifice plate, with the ones calculated by integration of the velocity profiles, the relative deviation obtained was 2.29% for exhaust fan velocity 1 and a decreased value of 1.80% for the exhaust fan velocity 2.

Furthermore, it is noted that the variation between volumetric flows measured on the four radius crossings were greater than the variation between the average value of radius crossings and the flow calculated using the orifice plate. Therefore, within the accuracy of the results obtained, the calculation of the flow through the differential device is valid.

### 3.1.2. Performance curves

Varying the flow on the installation, by controlling the vacuum pump and the installed valve, a graph was constructed with the static pressure measured in tapping number 2, as a function of airflow, for the first and second exhaust fan velocities. Figure 7 shows the curves obtained.

![Figure 7- Performance curves](image)

Working points can be determined by the intersection of the previous curves with the reference curve of IEC 61591 [2], $\Delta P=0.000375Q^2$. Assuming that experimental points behaviour comes close to a parabola, using the least squares method, $\Delta P = 138.98 + 0.7547Q - 0.0057Q^2$ is the equation that approximates the curve obtained for the first velocity (with $R^2 = 0.9853$) and $\Delta P = 206.97 + 0.4715Q - 0.0032Q^2$ (with $R^2 = 0.9952$) for the second one.

The flow rate corresponding to the working point is 225.63 m$^3$/h for the first velocity and 315.43 m$^3$/h for the second velocity. According to the Regulation 66-2014 [5], the fluid dynamic efficiency curve should be obtained only for the highest working exhaust fan velocity, the second velocity in this case. Figure 8 shows the FDE curve, corresponding the best efficiency point to a flow of 231.9 m$^3$/h.
Static pressure measurements and flows used to plot performance curves have an associated uncertainty. The expression used to compute the static pressure error was the following

\[ \text{Erro}_{P_{\text{stat}}} = \pm 2\sigma^2 \]  

Where \( \sigma \) is the standard deviation of the pressures and the 2 is a commonly used constant to represent a 95% confidence interval with 4 degrees of freedom.

As the flow is not directly measured, the expression above cannot be applied to calculate the error associated with the flow. \( dQ \) represents an error propagation related to pressure measurements, \( P_1 \) and \( P_2 \), which can be determined according to the following generic equation

\[ dQ = \left( \frac{dQ}{dP_1} \right)_{P_2} dP_1 + \left( \frac{dQ}{dP_2} \right)_{P_1} dP_2 \]  

Figure 9 illustrates the error bars associated with each reading.

Uncertainty concerning the static pressure is higher for higher pressures and uncertainty concerning to flow calculation is higher for smaller flows. The errors associated with the working points are ±0.63 m\(^3\)/h for the first exhaust fan velocity and ±0.46 m\(^3\)/h for the second. For the static pressure measurements, the errors are ±3.16 Pa and ±5.72 Pa for the first and second working points, respectively.

3.2. Visualization of the flow

Laser visualization allows better understanding of the difficulty in obtaining a fully developed flow, upstream of the measuring system.

Figure 10 represents the exhaust exit flow for the first exhaust fan velocity with an exposure time of 1/3200 sec. In this image, it can be clearly seen that the exit hood flow is completely asymmetric; being that approximately 70-75% of the airflow is deflected upwards, while only 25-30% follows an horizontal trajectory. For the second exhaust fan velocity, the results obtained were similar.

The jet formed in the outlet upper part results in an increased resistance to the fluid passage, consequence of the increasing shear stress on the wall. The occurrence of this phenomenon justifies why it is necessary a chamber with small spheres, since besides allowing an abruptly reduction of velocity, it canceled the tangential and radial velocity components.
4. INSTALLATION INTERNAL FLOW SIMULATION

The purpose of this part is to make a first evaluation of
the numerical methods available at the partner
compagny. The uncertainties associated with the
simulation of internal flows were analysed, evaluating
the mesh refinement influence for laminar and
turbulent flow solutions.
A second order discretization scheme was chosen for
the velocity in all simulations and of first order for the
pressure, being the iterative convergence required of
10E-6 for the normalized residue.

4.1. Code Verification

Code verification consists in verifying the absence of
errors in the program and allows the determination of
the numerical method convergence. A laminar flow
was simulated in a tube, which is a well-known case
and possible to solve analytically, allowing the
comparison with the computational results. It was
considered a steady-state flow of an incompressible
fluid and a Reynolds number of 1500 was used. For
the simulation was considered a flow in a tube with 1 m
diameter, 120 m length, smooth surface, an entry
velocity of 1 m/s and an atmospheric outlet pressure.
A no-slip condition was applied at the walls.
Five meshes were tested having 10800, 15700,
22920, 38500 and 67840 elements respectively.
VolRMS corresponds to the size of the representative
mesh element, related to the sum of all elements
volume. Using the maximum velocity at the centre of
the tube, at 120 m, the solution’s convergence can be
evaluated. In the limit, for a very refined mesh, the
solution tends to a value close to the analytical
solution, 2 m/s, as shown in Figure 11.
Considering a safety factor of 1.25, it can be verified
that the analytical solution is within the range for
\[ \frac{h_i}{h_1} = 1 \], being observed a first-order
convergence.

Figure 11- Velocity at the center of the pipe depending on
mesh elements size, laminar flow

Figure 12 represents pressure drop evolution in four
sections of the tube, more precisely between 90 and
100 m, 100 and 110 m, 110 and 115 m, 115 and 120
m.
If a variation between each pair of values is considered
the analytical solution is 0.02133 Pa/m. It can be seen
that the results tend to the analytical solution when
closer to the tube exit. However, the length of the tube
is not enough to obtain a fully developed profile and,
consequently, a linear pressure drop, being observed
again a first-order convergence.

As regarding shear stress, the conclusion is the same,
that it should be constant and equal to 0.00533 N/m²,
which is not confirmed by the data obtained. It is also
observed that the solution tends to approach the
analytical value as the distance to the exit decreases,
also being verified a first-order convergence.
4.2. Solutions Verification

For the simulation was considered a flow in a tube with 1 m diameter, 120 m length, smooth surface, a entry velocity of 1 m/s and an atmospheric outlet pressure. At the walls, a no-slip condition was applied. The turbulence models tested were the k-w and the SST, being Reynolds number considered of 10E5.

By default, irrespective of the location of the first grid points from the wall, the program automatically replaces these first points in order to use wall laws, so for turbulent simulations is always considered $11.06 < y+ < 50$.

Five meshes were tested with 22000, 31100, 44760, 69860 and 112640 elements for the k-w model and 22000, 32300, 47400, 81620 and 114560 elements for SST model.

$V_{m}$ corresponds to the size of the representative mesh element, related to the element size $h_{i}/h_{1}$.

Following the same procedure as for laminar flow, maximum velocity convergence was evaluated with mesh refinement. In Figure 13 it can be seen that the solution quickly converges to 1.166 m/s.

Using Colebrook’s equation, an approximation for the friction factor of 0.01799 was calculated, corresponding to a pressure drop variation of 0.00810 Pa.

The approximate shear stress corresponds to 0.002025 N/m$^2$, value that fits within the uncertainties obtained with the numerical data.

Proceeding similarly to the SST turbulence model, maximum velocity variations observed are minimal, being close to 1.174 m/s, independently of the mesh refinement used. This can be explained due to the selected distances for the first grid point, which are in the order of thousands.

Similarly to the model for the k-w, analyses with SST model shows that the pressure drop and the shear stress tend to the ones determined analytical, approaching these values as distances nearer to the pipe outlet are considered.

In sum, the convergence properties for the turbulent regime do not differ significantly from the laminar regime. Nevertheless, there is a difference in the uncertainty of the solution. For the k-w turbulent flow model the uncertainty found is one order of magnitude obtained, it should be noted it is an approximation based on experimental results, and numerical solutions obtained are also subjected to modelling errors associated with the choice of the turbulence model.

Nevertheless, it is noted that, just as in laminar flow, at distances close to the exit the pressure drop tends to get closer to 0.00810 Pa. This indicates that a longer length would be required to the tube, in order to obtain a fully developed profile, for the degree of mesh refinement considered.
above the laminar one, and for the SST model of two orders above.

4.3. Simulation of flow in exhaust hood box

After program verification, for typical and simple internal flow problems, a numerical tool was applied to the hoods’ study, especially as a first approach to the flow inside the exhaust box simulation, TL1-62. k-w was the turbulent model used and the working fluid was air, with 1.2 kg/m³ of density at 25 °C and with a Reynolds number of 5.3E4.

The geometry used corresponds to a TL1-62 exhaust hood box. Two models were used: the first representing only the hood box intending to evaluate the airflow behavior from the entrance area to the exit, without any kind of disturbance; The second model used was the hood box with the addition of the fan box, intended to simulate its influence on the flow deflection and consequent pressure loss. Two meshes were used, with 1.190.068 and 1.192.528 elements. The entry velocity used was of 0.66 m/s, in the outlet atmospheric pressure was verified and a no-slip condition was applied at the walls. Obtained flows are represented in Figure 15 and Figure 16.

It was visualized in both simulations that the velocity at the corners of the hood box is near zero, explaining why current hood configurations tend to have a more ergonomic shape, without corners.

5. CONCLUSIONS

In this work was studied and constructed a bench test for hoods characterization, in particular to establish performance curves and calculate the fluid dynamic efficiency. The bench test is to be applied in the industry and was developed in a partnership with Teka Portugal. Visualization of the flow in the exhaust outlet was made using a laser technique, with the purpose to understand the flow behaviour. It was verified a completely asymmetric flow at exit hood.

A numerical tool available in Teka was used to evaluate the accuracy of the simulation software tool used to simple internal flows and illustrated program’s capacity applied to the study of exhaust hoods with a concretized example. It was concluded that with the mesh refinement degree used, the typical accuracy of the solutions, in laminar regime, is of the order of thousandths with a first order convergence. In solution verifications, two turbulence models were evaluated and it was concluded that the typical accuracy of the solutions is of the order of hundredths for the k-w turbulence model and the order of decimal to the SST model, for the mesh refinement degree used.

6. REFERENCES