CFD design tool for industrial applications

Geanette Polanco
Simón BolívarUniversity, Caracas, Venezuela, gpolanco@usb.ve

Arne Holdo
Coventry University, Coventry, U.K., aa4083@coventry.ac.uk

Julio Longa
Simón Bolívar University, Caracas, Venezuela, jlongag@usb.ve

ABSTRACT

Geometry with sharp edges generally produces turbulent in the flow with high levels of noise generation that affects the flow characteristic. Simulations permit to establish predictions about the reductions of undesirable effects in the operational conditions of industrial processes. The example shown in this work consists of a generic air knife normally used in drying systems. Varying the geometry of a specimen in the simulation shows that a smoother contour reduces the turbulence level of the flow and its relational effects. This result suggests that the new design of air knives may include aerodynamic considerations to improve its performance.

Keywords: CFD, air knife, design tool

1. INTRODUCTION

Drying systems have an important role in a wide range of industrial processes. The efficiency covers the performance of each element inside. The air knife is the device most important in the drying line with the most possibility to be enhanced, (Paxton, 2008).

Normally, the air knives have a large body with different entries and one long a thin exit, (Turbotech, 2008). This generates a complex flow inside as well as a complex jet flow downstream the exit. Both situations will be examined by computational means using a 3D approach for the internal flow and a 2D approach for the external flow.

There are different cross sections with different dimensions. This obeys meanly to manufactures and the processes used to build them, see Figure 1.

The geometry studied in this work corresponds to one of the simplified cross section, as shown in Figure 2. The entrance is assumed to be a circular cross section of diameter of 80 mm and 20 mm long, located at one end of the body of the air-knife. The dimension of the air-knife body is 800 mm.

Figure 1: Common geometries of a air knives. Source: Sonic (2008), Air Knives (2008), Direct industry (2008) and Paxton (2008)

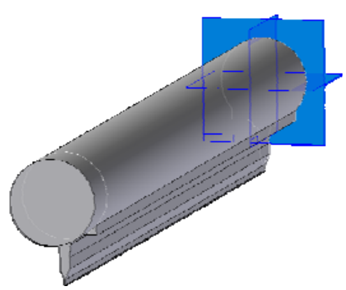

Figure 2: Simplified geometry. Named hereafter as original geometry

2. THE MODEL

TIME DEPENDENCY

The analysis is steady-state. No transient effects are taking into account for in any of the models.

THE FLUID

The fluid defined for the simulation was standard air, under the assumption of the uncompressible air flow.

THE MESH

Due to the high aspect ration of the nozzle located at the discharge location of the air-knife, a demanding mesh is needed to achieve low aspect ratios of the individual element of the domain. The characteristics of the grid used correspond to a structure non-orthogonal mesh in all cases. In order to despite any dependency of the result from the mesh used, a mesh sensibility study was also performed.

TEST CONDITIONS

The inlet velocity (54 m/s) is perpendicular to the inlet at the inlet depicting the fact of any change in direction of the flow due to the air supply system

BOUNDARY CONDITIONS

Boundary conditions for the internal body of the air-knife were assumed to be of constant velocity at the end-entrance of the air-knife and outflow at the gap of the nozzle. For the external study, a constant flow velocity in the gap and outflow in the borders around of the domain were selected.

TURBULENCE

In order to model the turbulent processes, the k-epsilon RNG and standard k-epsilon model were selected, for both internal and external flow of the air-knife respectively.

The industrial standard is the two-equation model which is an eddy-viscosity type model based on time averaged Navier-Stokes equations, which are sometimes termed the Reynolds Averaged Navier-Stokes (RANS) equations. The RNG k-epsilon turbulence model is a modification of the standard k-epsilon turbulence models, and the difference between these two models appears in the epsilon equation. The kinetic energy equations are identical. The RNG model is normally used in cases with flow close to walls or flow with recirculation zones, (Bird, 1966).

3. GOVERNING EQUATIONS

CFD is based on the numerical solution to the governing equations of fluid flow that express the conservation of mass, momentum and energy, (Launder and Spalding, 1974). The energy equation is not included in this work due to the assumption of incompressible flow. For a general 3D fluid the governing equations are:

(Mass)
$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho U_j)}{\partial x_j} = S_m \tag{1}$$

(Momentum)
$$\frac{\partial(\rho U_j)}{\partial t} + \frac{\partial(\rho U_i U_j)}{\partial x_i} = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_i}\left(\mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \rho\overline{u_i u_j}\right) + S_{Uj} \tag{2}$$

where $(i, j, k = 1,2,3)$ and $U_j =$ Components of the mean velocity in the x_j direction, $u_j =$ Components of the fluctuating velocity in the x_j direction, $h =$ Mean static enthalpy, $h^{'} =$ Fluctuating static enthalpy, $P =$ Pressure, $\rho =$ Density, $\mu =$ Dynamic Viscosity, $S_m =$ Source terms for mass (e.g. condensation, evaporation), $S_{Uj} =$ Source terms for momentum (e.g. body forces, buoyancy forces) and $-\rho\overline{u_i u_j} =$ Turbulent Stresses.

4. RESULTS

A selection of results obtained from the CFD calculation of each of the models is presented below:

INTERNAL FLOW

ORIGINAL GEOMETRY

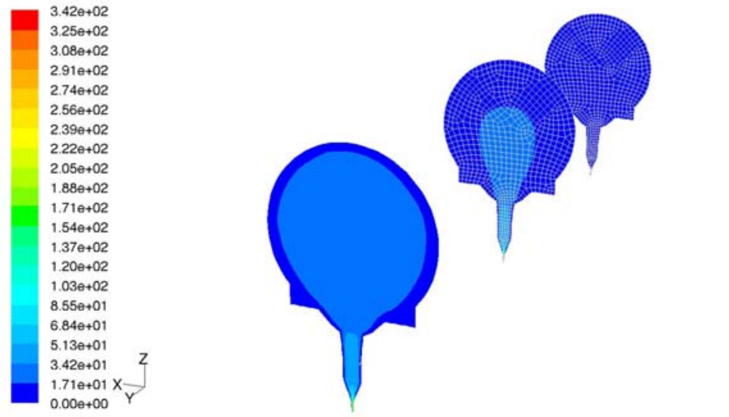

Figure 3: Contours of the velocity magnitude (m/s) for three different cross sections

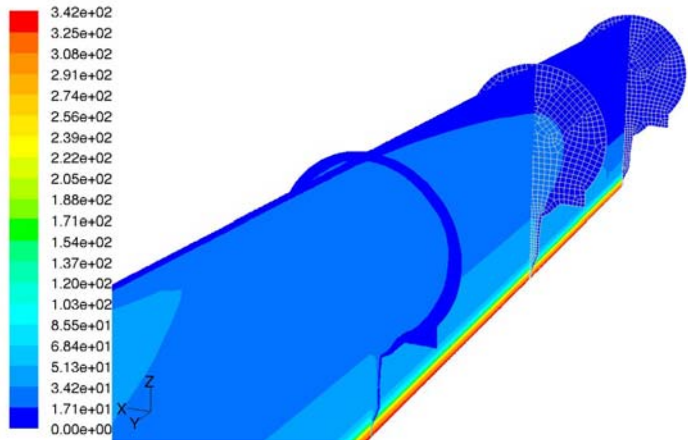

Figure 4: Contours of the velocity magnitude (m/s) for three different cross sections combined with a centre plane of the air-knife

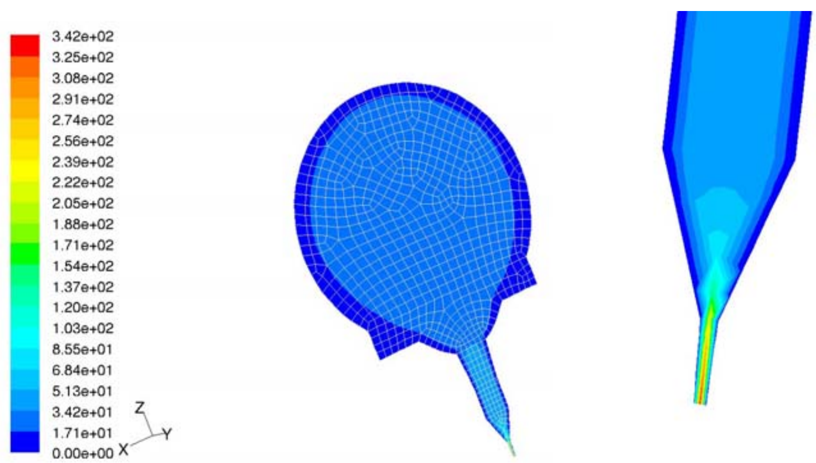

Figure 5: a) Contours of the velocity magnitude (m/s) at a cross section located at 0.5 m from the exit and the plane mesh. b) Detail of the contours of the velocity magnitude at the exit of the nozzle

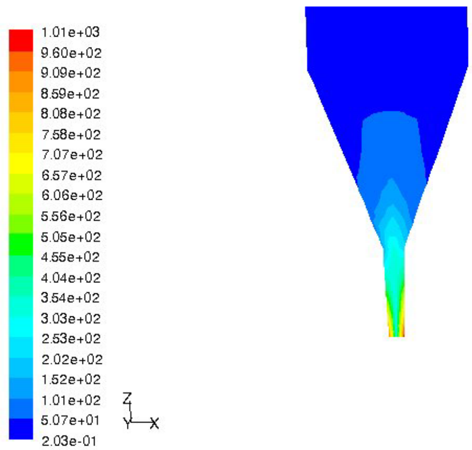

Figure 6: Contours of the turbulence kinetic energy (m^2/s^2) at a cross section located at the centre of the air-knife body

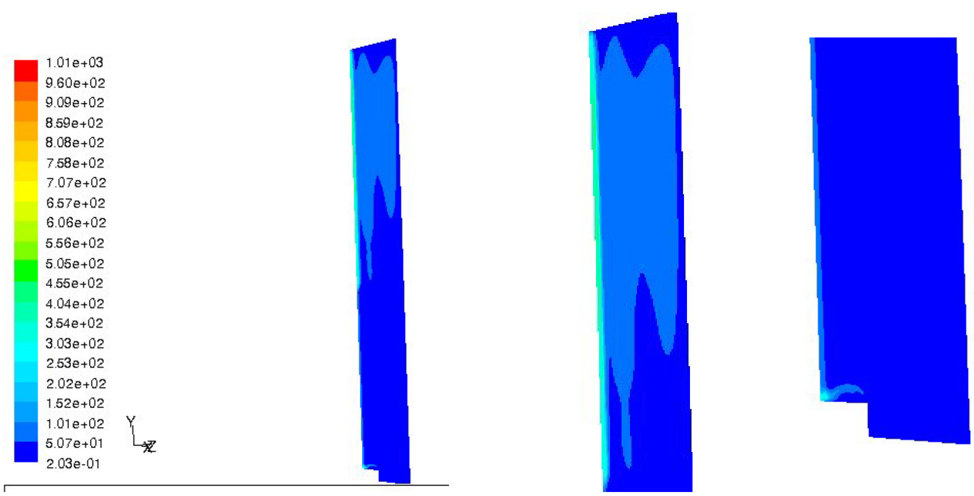

Figure 7: a) Contours of the turbulence kinetic energy (m^2/s^2) at a centre plane of the air-knife body b) Detail of the close end of the body c) Detail of the open end of the body

THE ENHANCED GEOMETRY

After performing the simulation of the current model, a new geometry was proposed, based on smoothing the shape, but keeping the dimensions of the exit gap and the diameter of the main body of the air-knife, as can be seen in Figure17.

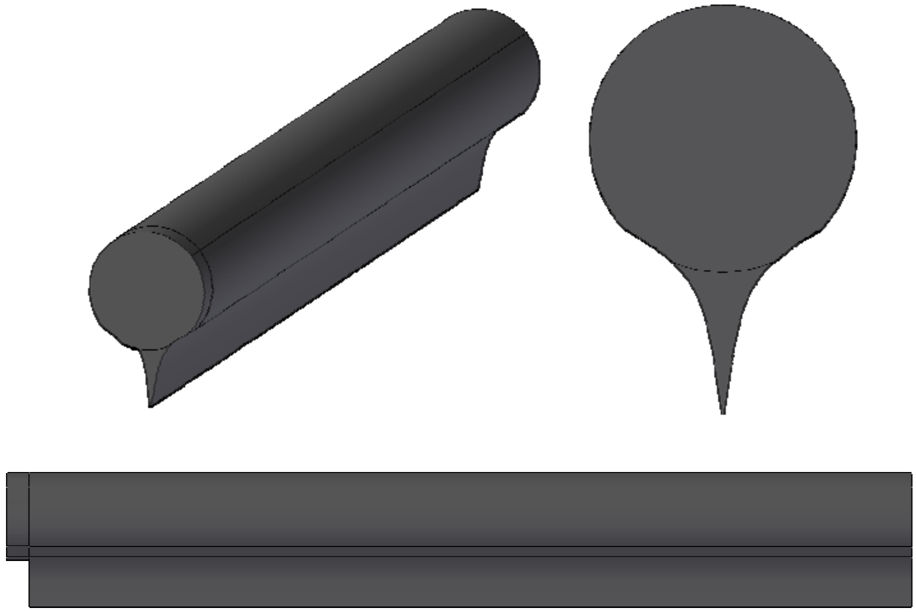

Figure 8: Proposed internal domain of the air-knife body

With the new geometry for the internal shape of the air-knife, new meshes were created and tested. The mesh aspect ratio in the gap is about of 2.35, whilst in the top is about 3.5.

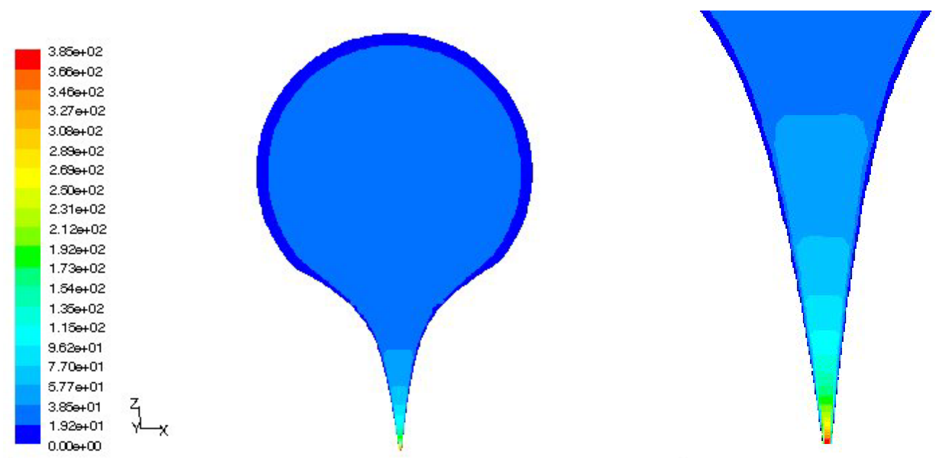

Figure 9: Contours of the velocity magnitude (m/s) for a cross section located at the centre of the air-knife body

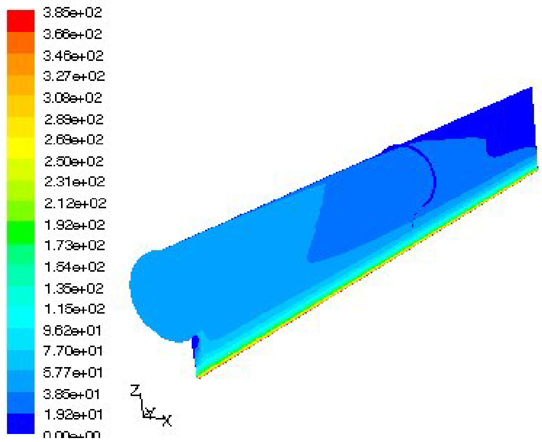

Figure 10: Contours of the turbulence kinetic energy (m^2/s^2)

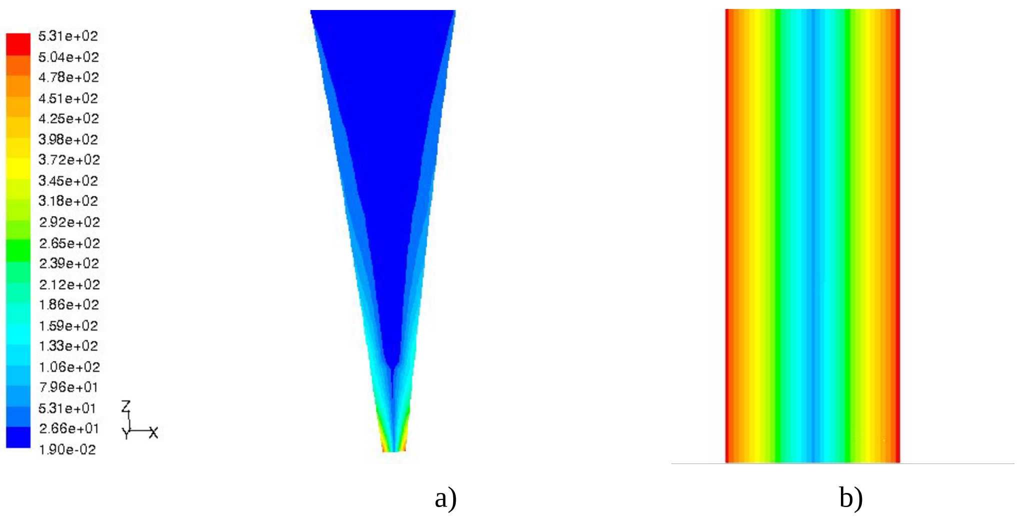

Figure 11: Contours of the turbulence kinetic energy (m^2/s^2) at a cross section located at the centre of the air-knife body b) Detail at the bottom of the geometry, at the exit of the nozzle

EXTERNAL FLOW

THE ORIGINAL GEOMETRY

The study of the air entrainment around of the body of the air-knife was performed using a 2D approach. It is important to observe the behavior of the flow around the solid shape and the impact of it on the turbulence.

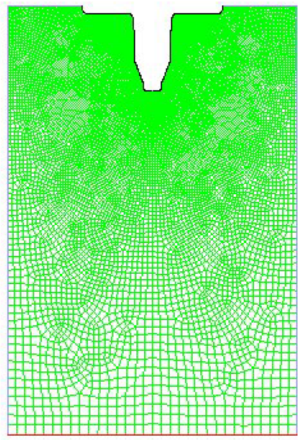

Figure 12: Plane mesh at the external part of the nozzle

The entrainment has a large effect on the momentum conservation of the jet, as it brings the energy necessary to keep the momentum constant in any cross section. In addition, to this condition it is known that any fluctuation or disturbance present in the surrounding will affect the characteristics of the jet boundary, so, in the absence of a smooth shape, the entrainment tends to be very disturbed and then the jet boundaries far from the nozzle makes a form of flapping motions.

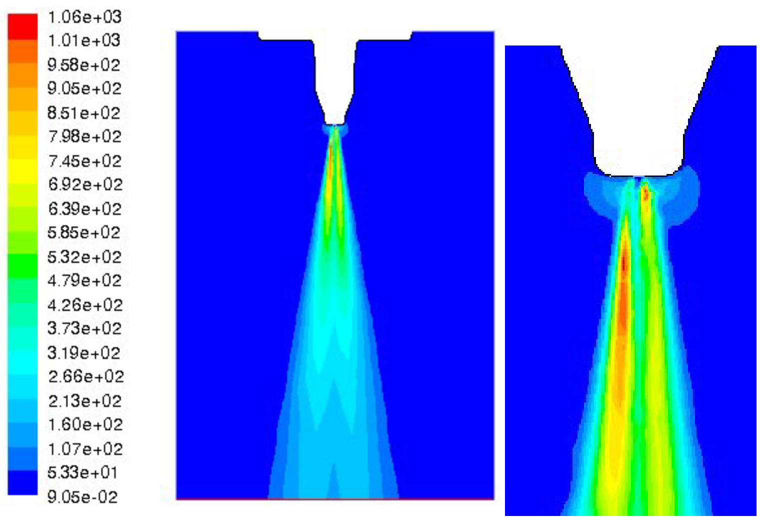

Figure 13: Contours of the turbulence kinetic energy (m^2/s^2) around and downstream of the exit of the nozzle, for longer domain

THE ENHANCED GEOMETRY

The proposed geometry for the external part of the air-knife consists of the creation of a new curve, which substitute the whole external shape of the nozzle and the connection with the cylindered body. This curve allows a smoother connection between the external body and jet.

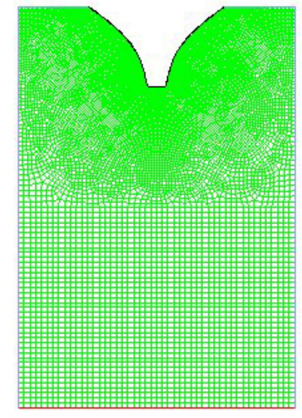

Figure 14: Plane mesh at the external part of the nozzle, for the proposed geometry

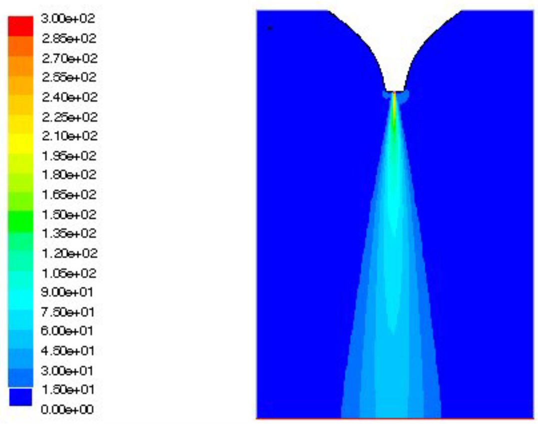

Figure 15: Contours of the velocity magnitude (m/s) downstream of the exit of the nozzle, for the proposed external shape

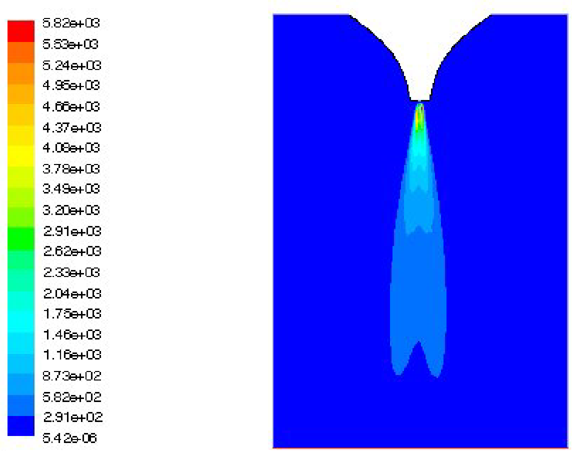

Figure 16: Contours of the turbulence kinetic energy (m^2/s^2) around and downstream of the exit of the nozzle, for longer domain

5. DISCUSSIONS

From the observation of the Figure 1 to 11, it can be discussed that no impinged jet is observed at the opposite end of the air-knife body, which means the actual length is enough to distribute most of the kinetic energy at the inlet inside the body toward the nozzle exit. The velocity profile obtained allows the identification of slower zones close to the walls and toward to the opposite end of the body of the air-knife. No major recirculation in the transversal cross section along the axis is presented. The new proposed internal shape produces a more natural flow of air inside the body of the air-knife. The velocity level achieved at the exit shape is almost the same for the both geometries. The production of turbulent kinetic energy, the dissipation rate and the turbulent intensity for the proposed internal shape were reduced in all cases respect to the original geometry.

The jet velocity contours at the exit of the nozzle produces by its interaction with the external shape of the air-knife, see Figures 13. The improved shape helps to reduce the perturbations in the surroundings and consequently it helps to obtain a more stable jet, as shown in Figures 15 and 16.

6. CONCLUSIONS

The use of CFD as a design tool allows showing the influence of the aerodynamic surface on the flow characteristic. Particularly in the case of an industrial device such as an air knife, this influence on internal and external flow configuration enhances the turbulent phenomenon presented, and therefore the design of new air knife must consider this aspect as a very important to improve the overall performance of the air knife and the whole drying system. The testing capability of new designs using CFD is an unlimited resource.

REFERENCES

Air Knives (2008), http://www.air-knives.com/, April, 2008

Bird, R.B., Stewart, W.E., and Lightfoot, E.N. (1966). “Transport Phenomena”. John Wiley & Sons, New York.

Direct industry (2008), http://www.directindustry.com/industrial-manufacturer/air-knife-73506.html

Launder, B.E. and Spalding, D.B. (1974), “Lectures in mathematical models of turbulence”. Academic Press, New

Paxton (2008), http://www.paxtonproducts.com/products_airknives.php, April, 2008

Sonic (2008), http://www.sonicairsystems.com/air_knife_drying.php, April, 2008

Turbotech, (2008). http://www.republicsales.com/products/air-knife-drying-systems/republic-airknife/, April, 2008

Authorization and Disclaimer

Authors authorize LACCEI to publish the paper in the conference proceedings. Neither LACCEI nor the editors are responsible either for the content or for the implications of what is expressed in the paper.