13th International Symposium on Particle Image Velocimetry – ISPIV 2019
Munich, Germany, July 22-24, 2019

Investigation of a large angle diffuser in the steady stall regime via 3D3C mean vector field statistics

Bastiaan Bijvoet[1*], Dr. Jens Kitzhofer[2], Mircea Dinulescu[2]

[1] Delft University of Technology, Fluid Mechanics, Delft, The Netherlands

[2] Apex Group, Research B.V., Voorburg, The Netherlands

[*] bastiaan.bijvoet@apexgroup.eu

Abstract

In heavy duty industry, such as the petro-chemical industry, often the efficiency of fired heaters is increased with air-preheaters. On their cold side, the flow is driven with forced draft centrifugal fans. Due to space limitations the transition from the outlet of the fan (small cross section) to the inlet of the air preheater (large cross section) needs to be overcome in a small distance. Naturally there is the requirement for a low total pressure drop, and additionally a large degree of uniformity at the diffuser outlet, i.e. the air preheater inlet.

This paper aims to extend the knowledge on flows in large angle diffusers. These are often the most unstable component in fluid mechanical systems, as their performance, expressed in terms of pressure recovery and flow distribution, can vary a lot. Some 60 years ago, Kline and Fox (1958) were the first to come up with a predictive tool for diffuser performance.

Their study considers a uniform velocity distribution at the inlet of the diffuser. This is not the case in industrial applications as the velocity field emerging from centrifugal fans is non-uniform, and can have secondary flow structures such as swirl. Another discrepancy between literature and industry, is the resistance just downstream of the diffuser, in the form of the air preheater. The poor performance itself of fan-diffuser systems, and the discrepancies with literature have led to this research.

A windtunnel was built with a diffuser shaped measurement section. Laser optical measurements were performed to characterize the flow, also pressure measurements were performed. With knowledge of the flow, in terms of velocity and pressure, it was possible to judge the performance of the system. The measurements have the goal of extending the knowledge on diffuser flow itself, and to define the boundary conditions for subsequent CFD analyses.

1 Background

Diffusers are ducts that have an outlet that is larger than the inlet. Their goal is to slow down a flow and with that, to increase the fluid pressure. This increased pressure will for this application drive the flow through the equipment downstream of the diffuser. In other words, diffusers have the goal of converting kinetic energy into pressure. Another requirement is that the flow at the diffuser outlet is uniformly distributed as downstream equipment is most often designed for uniform inlet conditions.

1.1 Flow regimes

It was first described by Kline and Fox (1958) that four easily distinguishable flow patterns can occur in a diffuser. At high Reynolds numbers, what flow pattern will be present is mostly a function of the diffuser geometry, see Figure 1(b). Here it can be seen that for long, slender diffusers, *no stall* occurs. In these there are low losses, however they can not deliver any substantial pressure rises.

At increasing angles diffusers can show *transitory stall* and *steady stall* where respectively the high velocity jet has a position that alternates between the top and bottom wall (as in Figure 1(a)), or has a position fixed to either one of the two. The highest pressure recovery is achieved in the transitory stall regime, indicated by the line $C_{p,max}$ in Figure 1(b), see Section 1.2. In diffusers of even larger angles, the jet follows no wall. Regions of recirculation exist above and below the jet. Note that the symbols used to describe the geometry of the diffuser are shown in Figure 1(a).

1.2 Performance

For this research, the performance of a system consisting of a fan and diffuser (that leads up to a plate-type heat exchanger) will be quantified using five performance parameters, these are explained in the oncoming sections.

Flow rate: The flow rate of the system is seen as an output, as the fan has a fixed speed[1]. It is a good indicator for a reduced pressure drop (of the windtunnel as a whole), that future improvements to the system are intended to induce. Changes made to the system should result in a higher flow rate[2].

Kinetic energy flux factor at the diffuser outlet: Generally in engineering, it is desirable to express field or volume quantities with a single quantity, for easy understanding and comparing. For describing the non-uniformity of a velocity field, the kinetic energy flux factor (α) is used. It is defined as the flow of kinetic energy of a given velocity profile, normalized with the theoretical minimum flow of kinetic energy for fixed flow area and mass flow. This can be written as in Equation 1. Naturally α cannot be below one, át one, the corresponding flow has a fully flat velocity profile (that does not meet the no slip condition).

$$\alpha = \frac{1}{A}\int_A \left(u/\bar{u}\right)^3 \mathrm{d}A \tag{1}$$

The uniformity of the velocity field at the diffuser outlet, which for the scope of this work is the inlet into a heat exchanger, is very important to the performance of both the diffuser and the heat exchanger: On the diffuser side it directly influences the static pressure recovery. A high kinetic energy flux factor at the diffuser outlet (α_2) indicates that still a large amount of kinetic energy is present in the flow that should have been converted into pressure. Regarding the air preheater, these are generally designed to cope with a uniform mass flow over their inlet area. A high α_2 will thus lead to a decrease in performance of the heat exchanger, as the separate channels of the heat exchanger will experience flow rates that deviate from their design flow rate.

Static pressure recovery coefficient: Historically, the increase in static pressure over the length of the diffuser, was seen as the most important performance parameter for diffusers. The C_p indicates to what degree the flow of kinetic energy at the diffuser throat is converted into pressure. It is a measure for how effective a diffuser is. A widely accepted definition for C_p is given in Equation 2, for the scope of work this one dimensional expression is too simplistic. In reality a non-uniform field exists for the velocity and pressure, and due to nonlinear relations between the velocity, pressure and the C_p, the use of scalars will lead to errors.

$$C_p = \frac{\Delta p}{\frac{1}{2}\rho u^2} \tag{2}$$

See Equations 3a and 3b: For the term of the pressure rise, the difference is taken between the mass flow rate weighted mean pressure at diffuser throat and outlet. Also for the dynamic pressure, the denominator in Equation 2, a mass flow rate weighted average must be used. It is convenient that the kinetic energy flux factor is already used as a performance parameter, as it can be used to obtain a more simple expression for the mass flow rate weighted mean dynamic pressure. Note that $\bar{u}_1$ represents the mean velocity in the diffuser throat ($\bar{u}_1 = Q/A_1$). The kinetic energy flux factor is as defined in Equation 1.

$$\Delta p = \frac{1}{Q}\Big(\int_{A_2} pu\,\mathrm{d}A - \int_{A_1} pu\,\mathrm{d}A\Big) \tag{3a}$$

$$C_p = \frac{\Delta p}{\frac{1}{Q}\int_{A_1}\frac{1}{2}\rho u^3\,\mathrm{d}A} = \frac{\Delta p}{\frac{1}{2}\rho\bar{u}_1^2\cdot\alpha_1} \tag{3b}$$

As a point of discussion; the definition chosen for C_p, differs from the one used in literature, resulting in a weak basis for comparison between the measurements and predictions. On the other hand, as the physics (non-uniform velocity profile) in the system at hand are different than the physics assumed in literature (uniform profile) there was never going to be a good comparison, and so it is acceptable to have different definitions, as this results in a better description of the physics (which contradictorily could lead to a bétter basis for comparison).

[1] Although it is somewhat sensitive to changes in the frequency of the electrical power grid.

[2] Low Mach numbers are expected and so either the mass flow rate or the volume flow rate can be used.

Static pressure recovery efficiency: Complementary to the effectiveness C_p, an efficiency is used as performance parameter. Normalizing the static pressure recovery coefficient with its ideal value (that holds for inviscid flow), gives the static pressure recovery efficiency, defined as in Equation 4.

$$\eta = \frac{C_p}{C_p^*} = \frac{C_p}{1-(A_1/A_2)^2} \tag{4}$$

Dissipation in diffuser: For the time rate of energy dissipation in the diffuser, $\dot{W}_{viscous}[W]$, the equations of 5 were used. These are a direct result of conservation of energy. With the PIV measurements only mean velocities would be measured, and so a slight under-prediction will be found here for the flows of kinetic energy[3]. Note that the subscripts 1 and 2 respectively refer to the diffuser throat and diffuser outlet.

$$\dot{W}_{viscous} = \Delta\dot{E}_{kinetic} + \dot{W}_{pressure} + \Delta\dot{E}_{elevation} \tag{5a}$$

$$\Delta\dot{E}_{kinetic} = \int_{A_1} \frac{1}{2}\rho\bar{u}\left(\bar{u}^2+\bar{v}^2\ +\bar{w}^2\right)\mathrm{d}A - \int_{A_2} \frac{1}{2}\rho\bar{u}\left(\bar{u}^2+\bar{v}^2\ +\bar{w}^2\right)\mathrm{d}A \tag{5b}$$

$$\dot{W}_{pressure} = \int_{A_1} p\bar{u}\,\mathrm{d}A - \int_{A_2} p\bar{u}\,\mathrm{d}A \tag{5c}$$

$$\Delta\dot{E}_{elevation} = \int_{A_1} \rho g z\bar{u}\,\mathrm{d}A - \int_{A_2} \rho g z\bar{u}\,\mathrm{d}A \tag{5d}$$

2 Experimental set-up

A scaled-down (1 : 5) experimental set-up was built. This closed loop wind tunnel comprises of a $630\,W$ single inlet centrifugal fan, a straight walled diffuser and a geometric model of a plate type heat exchanger. In this heat exchanger model, no heat transfer occurs, its goal is to simulate the resistance of an actual heat exchanger. It does it by forcing the flow to enter horizontal channels that have a height of $9\,mm$, these channels are spaced by $9\,mm$ thick wooden plates and so effectively the average velocity is doubled.

The diffuser has a fixed width of $254\,mm$ and has a throat of $W_1 \times W_2 = 322 \times 254\,mm^2$. The area ratio is $W_2/W_1 = 5$, the length is $L/W_1 = 4$. Literature suggests that the flow regime in this diffuser would be that of *steady stall*, see Figure 1(b). In Figure 2 an impression is given of the windtunnel that was built, Figure 3 gives a top view with all relevant dimensions.

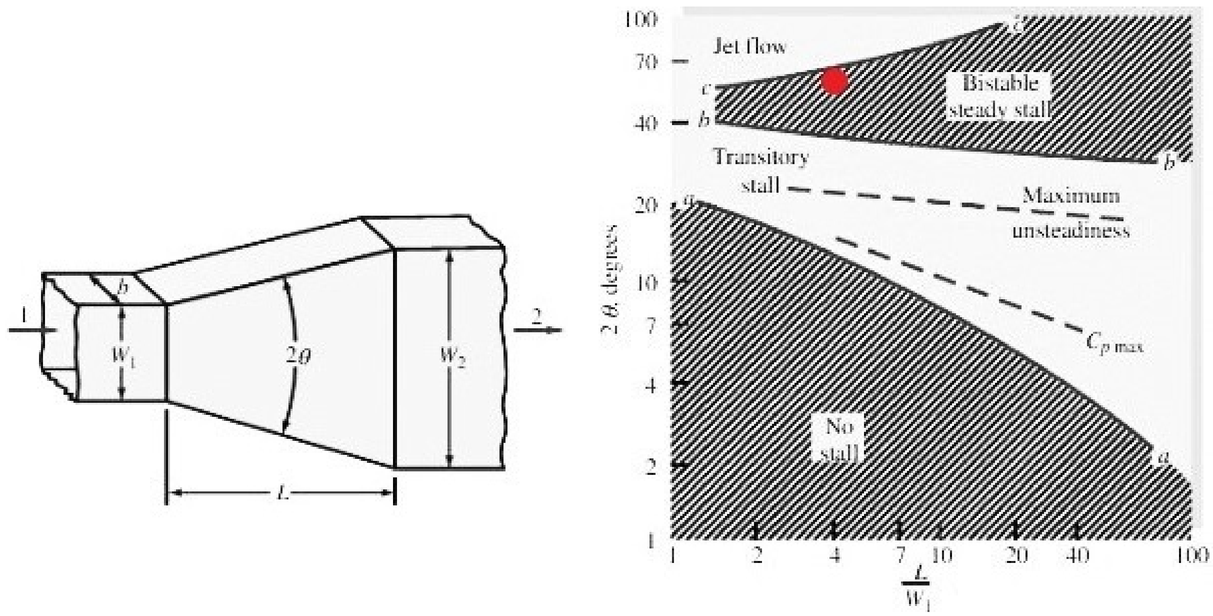
The measurement section is made of transparent perspex, it extends from the fan outlet, through a short straight section and the diffuser to some distance into the separate channels. With optical access into these channels it is possible to accurately measure the distribution of flow over the height of the air preheater model. The remaining ducts are made out of coated MDF wood.

3 Measurement equipment

LDA: For the Laser Doppler Animometry 'Dantec Dynamics' equipment was used, more specifically the 'FlowExplorer' in conjunction with the 'F60 BSA flow processor'. The FlowExplorer is a single unit housing all the emitting and receiving optics required for measuring two velocity components. The burst spectrum it puts out, is processed in the processor. The software used to monitor and control the laser, sensor and the traverse was 'BSA flow software'. All relevant specifications of the LDA system are listed in Table 1. With the $300\,mm$ focal length, the LDA system was able to measure across the whole width of the measurement section as its internal width plus one wall thickness was $262\,mm$.

Multiplane PIV: For the PIV measurements, the 'Litron Lasers Bernoulli' laser was used, the $4.3\,MP$ 'Dantec Dynamics FlowSense EO' camera was used. Table 1 also gives the specifications of the PIV system used. With the PIV system at hand two velocity components in a two-dimensional region can be measured, i.e. a $2D2C$ description of the mean velocity field. To have a $3D3C$ description of the mean velocity field, the following method was used:

[3] As future improvement, turbulence data from LDA measurements could be used to predict magnitude of the fluctuations, and find a better estimate for the kinetic energy flows.



(a) The symbols used to describe the geometry of a straight walled diffuser, they are as in the work of Frank White (2011).

(b) The expected flow regime depends on the geometry of the diffuser.

Figure 1:

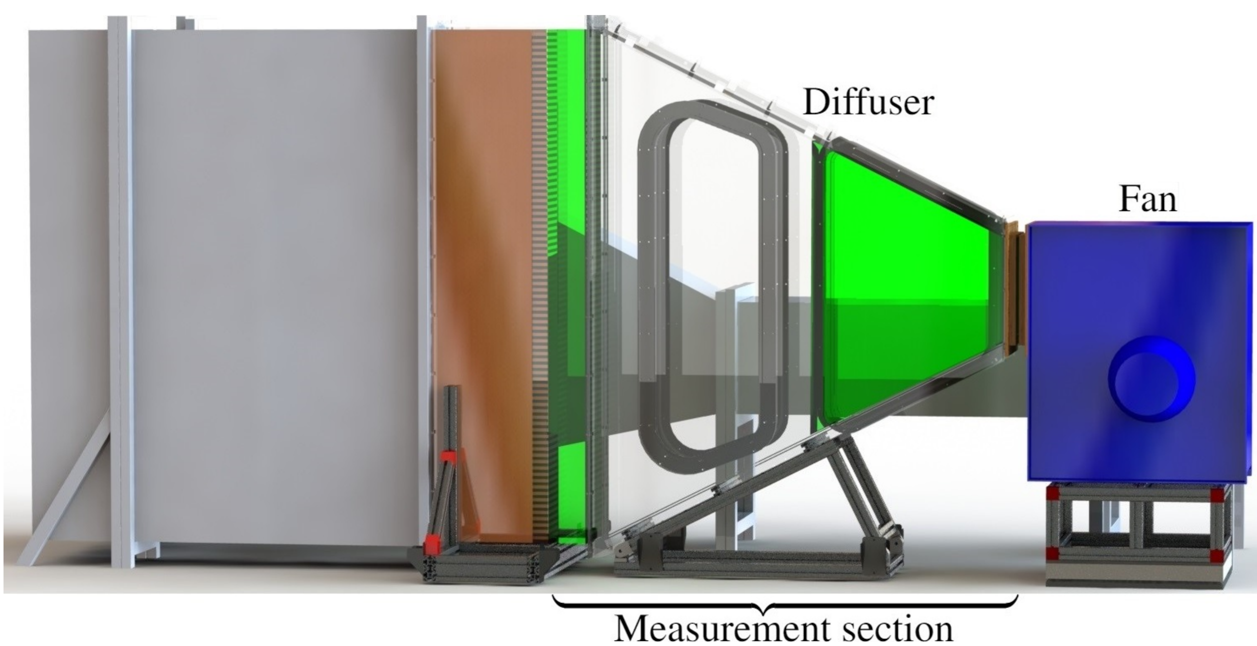


Figure 2: Impression of the test facility built, the return duct can be seen through the transparent measurement section (length from left to right $\approx 4m$). Downstream of the fan (blue), are the diffuser and the channels (brown). The grey duct is the return durct. The sections marked in green are of particular interest.

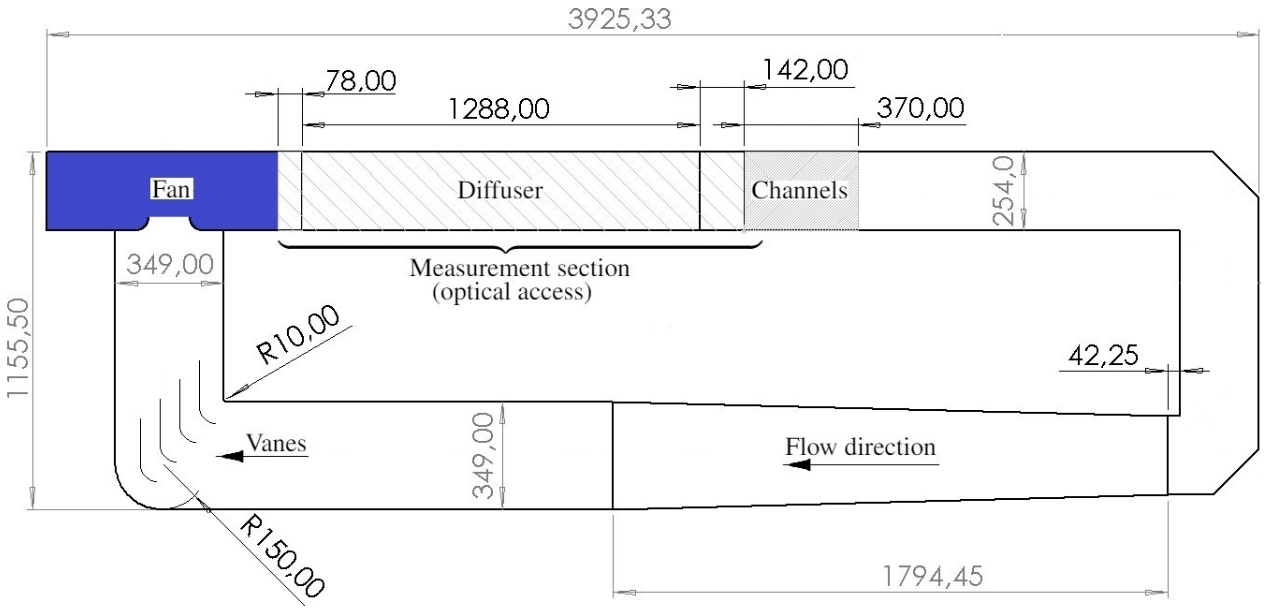


Figure 3: Top view with relevant dimensions of the windtunnel. The transparent measurement section extends from the fan outlet, to 50 mm into the channels that simulate an air preheater. As in Figure 2, the fan is in blue.

Table 1: System specifications & settings of the LDA and the PIV systems

LDA	
Laser power	500 mW
Beam spacing	63 mm
Fringe spacing	1.33 μm
Measurement volume	$42 \times 41 \times 210 \mu m$
Stop criterium	45 s
Average data rate	1.3 kHz

PIV	
Laserpulse energy	$E = 200\,mJ$
Wavelength	$\lambda = 53\,nm$
Resolution	120 - 360 $\mu m/pix$
Interrogation area min. size	$16 \times 16\,pix$
Max. particle displacement	13.9
Number of double images	200
Frame rate (double im.)	10 Hz

Multiple horizontal planes (blue, in Figure 4) were measured. In each of these planes the two in-plane velocity components were found, combining the data of all these planes gave a $3D2C$ velocity field, $\bar{u}$ and $\bar{v}$ in many planes, that together span a volume. The same was done for vertically oriented planes (red), and so again another $3D2C$ velocity field was found, now with the $\bar{u}$ and $\bar{w}$ components. In order to combine these data sets, they firstly had to be interpolated onto a grid that is exactly the same for both. Then, they could be combined; for the streamwise velocity component that existed in both sets, the mean was taken. The other velocity components were copied.

A traverse was used to accurately position the camera and the laser. These were mounted fixed in relation to one another, this had the benefit that only once the system had to be calibrated, even though many planes were measured. As long as the two stay fixed in relation to one another, there was no need for recalibrating. To conclude, this *multiplane PIV method* can only be applied to the true mean velocity field. If due to a too short measurement time, no true mean is obtained in at least one of the two data sets, the two are not compatible.

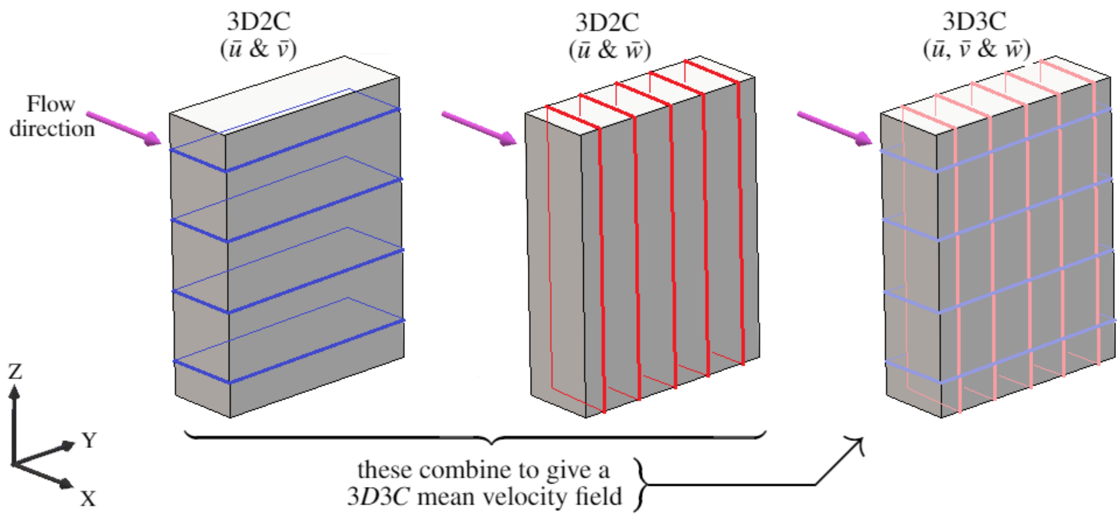


Figure 4: Schematic overview of the *Multiplane PIV* method, in which conventional PIV measurements are used to obtain a three dimensional velocity field with all three components.

Wall pressure measurements: For measuring the pressure throughout the windtunnel wall pressure measurements were performed[4]. Over a single cross-section the pressure is often nearly constant and so it is acceptable to limit to wall pressure measurements. With hindsight, this statement holds for all positions downstream of the plate stack, not for the cross-sections of the diffuser throat and outlet. At these positions, respectively 24 and 28 wall pressures measurements were taken, spread out along the perimeter, and so still good insight of the physics was obtained.

4 Results

Visualization with flexible strips: An effort was made to do a large scale visualization with plastic strips that follow the flow. At relevant positions, these were attached to the inner wall, or to a steel rod of $4\,mm$ thickness placed in the vertical centerplane of the channel. The channel was illuminated with high power LED's and pictures were taken. The result a shown in Figure 5. In the image the plastic strips are not adequately visible with the naked eye and so they were highlighted with yellow and red: The yellow dot indicates the position were the strip is attached, the red stripe indicates the direction the strip has at thát moment, most strips had a fixed orientation. Those at positions $1, 2, 3, 4$&5 díd, those just below the centerline, connected to the steel rod (in between 3 and 4) showed large fluctuations, and those in the rectangle at position 6 also showed fluctuations, albeit to a much lesser extent. From the image image it can be concluded

[4]It would have been highly impractical to measure the pressure in the bulk of the flow, as this would have yields using a pitot tube. This is highly impractical in the ever-changing three-dimensional flow field, as the pitot tube must be aligned with the flow to give a reliable measurement.

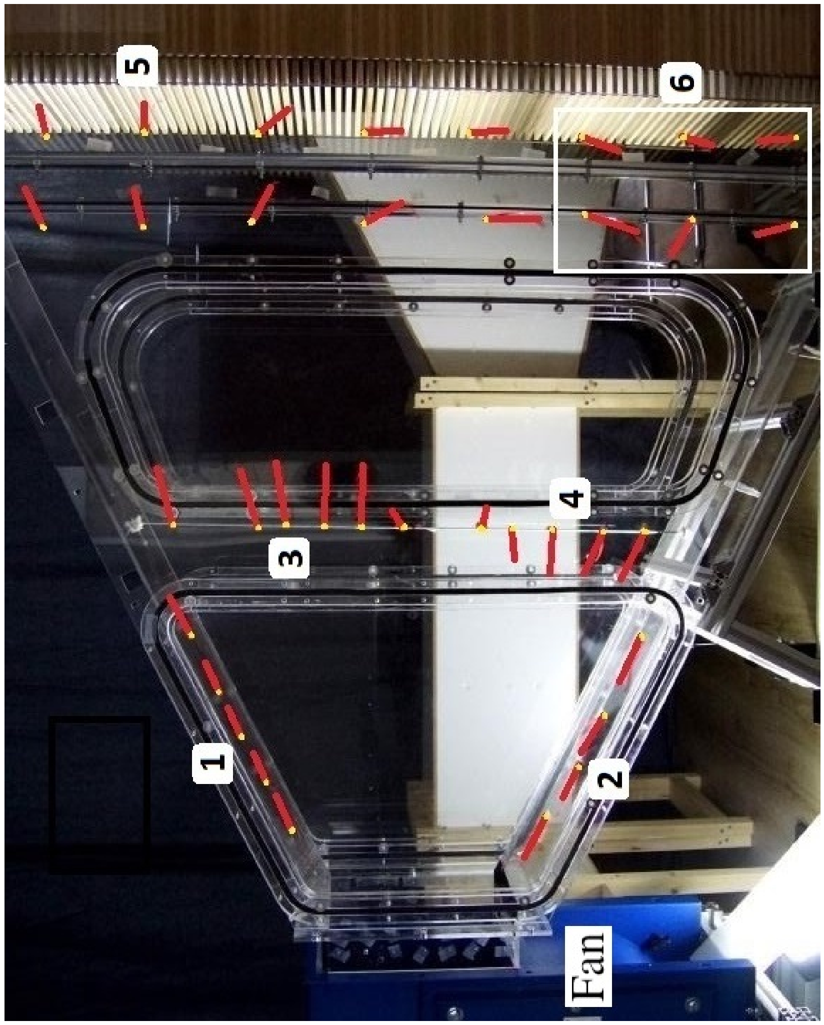
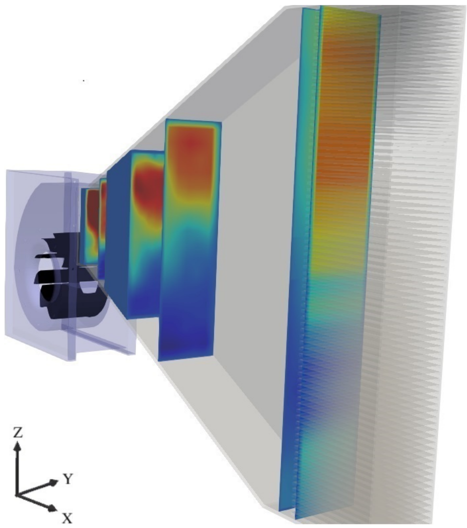


Figure 5: A side view of the measurement section of the windtunnel, with plastic strips indicating the local direction of the flow. The strips are highlighted here in red, the *base* of the strip is marked in yellow. This image was mirrored along its vertical centerline (and does not agree to other drawings of the windtunnel), to meet the convention of letting the fluid flow towards the right-hand side.

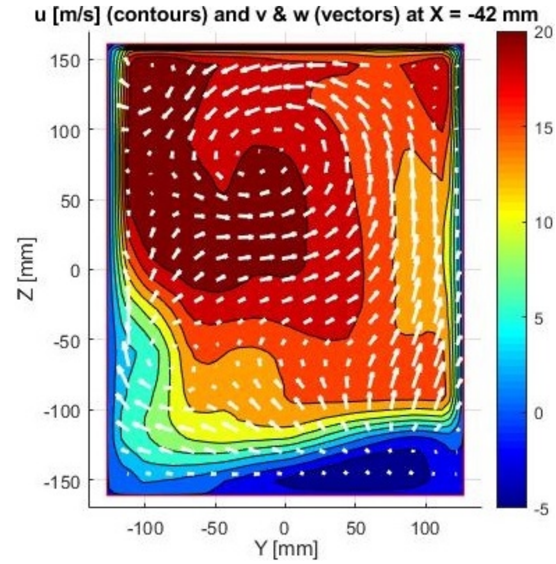
that a jet exists, it is attached to the top wall. Underneath the jet, there is a large region of recirculation, spanning the whole length of the diffuser. It cannot be seen in Figure 5, however from the same visualization it was concluded that the jet has a swirling motion, as the strips attached to the top surface were angled accordingly.

Velocity measurements: At several positions the velocity field was measured, at all points where the mean streamwise velocity component $\bar{u}$ was measured, it is shown in Figure 6(a). Looking closely, it can be seen that at two cross section just downstream of the diffuser, $\bar{u}$ is given and that over the distance of only $100\,mm$ that is in between the two, the velocity drastically increases. This is caused by the air preheater model with a void fraction of $1/2$. In the second cross section the flow has entered the channels, and so effectively the flow area is halved, doubling the average velocity. In Figure 6(b), the velocity field in the straight section in between the fan and the diffuser is shown. The velocity field shown derives from the *multiplane PIV* method (see Section 3). The point of view is such that the reader is looking into the fan. Near the top left a jet can be seen, that has a swirling motion. This swirling motion was also observed in the above mentioned visualization.

Pressure measurements: It was found that a pressure rise of $86.0\,Pa$ occurs over the length of the diffuser. This number is based on the mass flow rate weighted average pressure at the in- & outlet cross section of the diffuser, see Equation 3a.



(a) Results of all velocity measurements, colouring is linear to the X-component of the fluid velocity. The color scaling is consistent with that of Figure 6(b).



(b) Velocity profile at the diffuser throat ($42\,mm$ upstream of where diffuser starts), a jet can be seen, near the top left. Also the jet has a swirling motion. The length of the vectors that give $\bar{v}$ and $\bar{w}$, is such that the longest vectors corresponds to a velocity magnitude of $7\,m/s$.

Figure 6: Some of the measurement results.

5 Analysis

With the data gathered during the velocity and pressure measurements, all the performance parameters that were set in Section 1.2, could be derived. These, and some more parameters are given in Table 2. According

Parameter	**Symbol**	**Value**
Volume flow rate	Q	$1.16\,m^3/s$
Reynolds number	Re	$2.7 \cdot 10^5$
Static pressure recovery coeff.	C_p	0.43
Static pressure recovery efficiency	η	$45\,\%$
Kinetic energy flux factors	α_1/α_2	$1.64/4.53$
Dissipation in diffuser	$\dot{W}_{viscous}$	$90.1\,W$
Hydraulic fan power	$\dot{W}_{fan}$	$243.3\,W$
Fan electrical power	$\dot{E}_{fan}$	$630\,W$

Table 2: Overview of the performance of the fan/diffuser system.

to literature (Figure 1(b)), the flow regime in the diffuser at hand should be *fully developed two-dimensional stall*. It was found that indeed, the system operates in this regime. The swirling jet sticks to the top wall of the system. Furthermore it was found that the flow has a very non-uniform distribution at the outlet of the diffuser. LDA measurements inside the channels showed that no less than $84\,\%$ of the mass flow passes through the top half of the air preheater model.

In the fan, the rate of energy transferred to the flow is $243.3\,W$, taking into account the rate of pressure work and terms of kinetic energy and potential energy (the fan outlet is higher than the inlet). The electrical power input into the fan was $630\,W$ and so it was concluded that the fan is performing very poorly. The Air Movement Control Association (AMCA (2007)) suggests that if the velocity and pressure of the flow emerging from a fan, are measured too close to the fan outlet, the ratio of dynamic to static pressure can be different to what is predicted in the fan specifications. This was the case, but it does not explain the low efficiency of the fan. That most likely has to do with the limited length of straight duct upstream of the fan. Even though guiding vanes with straight tails were used (see Figure 3) to straighten the flow, probably a swirling flow enters the fan, which according to literature can explain the poor fan performance. It is suggested that the losses related to this unfavorable fan inlet geometry, can lay in the range of $200\,Pa$, which is consistent with the observed power defisit.

6 Conclusion & future work

It is hard to predict the performance of a fan/diffuser system as in previous work usually a uniform velocity field and no resistance downstream of the diffuser is considered. It was predicted (Frank White (2011)) that the diffuser in the windtunnel built, would be stalled and that no transient effects would be observed. This was confirmed by a preliminary visualization, and by quantitative measurement with LDA and PIV.

No literature was found that predicts the pressure recovery for diffusers with a non-uniform inlet velocity profile. However comparing the measured $C_p = 0.43$ to tables in the work of Frank White (2011) shows that the diffuser in the windtunnel performs better than expected. It must be kept in mind here, that in literature a one dimensional expression for C_p is used, whereas the C_p that was found, is based on a surface integral (see Section 1.2).

Future work should aim at methods to increase that pressure recovery, increase the flow uniformity and to reduce losses. A starting point is inserting guiding vanes in the flow, their leading edges should be near the diffuser throat. Near the diffuser throat, a $3D3C$ description of the mean velocity field is present. This can be used to properly design the guiding vanes in terms of their profile and their orientation.

Besides experimental work, a computational study can be performed now that the inlet boundary conditions of the diffuser are known. In conjunction with the windtunnel, future CFD work can be validated.

References

AMCA (2007) Publication 201-02 02

Frank White (2011) *Fluid Mechanics*

Kline S and Fox R (1958) *Optimum design of straight-walled diffusers*. Dept. of Mechanical Engineering Stanford University, Stanford Calif.