Improved modelling of wet scrubbers

(2012/055)

Final Report submitted to Sugar Research Australia

Anthony Mann

Queensland University of Technology

May 2017

SRA Project Code	**2012/055 (QUT055)**
Project Title	Improved modelling of wet scrubbers
Key Focus Area in SRA Strategic Plan	Milling efficiency and technology (KFA5)
Research Organisation(s)	Queensland University of Technology
Chief Investigator(s)	Dr Anthony Mann Dr Floren Plaza Dr Phil Hobson
Project Objectives	The project seeks to gain an improved understanding of the flow processes in wet scrubbers. These flow processes include but are not limited to: • Flue gas and droplet flows; • Agglomeration of droplets; • Flow of bulk water; and • Interaction between dust particles and water droplets. With the improved modelling capability, existing scrubber designs can be simulated with the aim of determining design modifications for improved scrubber performance.
SRA measures of success for Key Focus Area (from SRA Strategic Plan)	Adoption of improved or novel milling processes and technology

Contents

Executive Summary

Recent record high levels of electricity export from sugar factories have corresponded to record low levels of dust emissions from sugar factory boilers. One of the main reasons for low emissions is the increased use of wet scrubbers for removing dust from boiler flue gas.

While wet scrubbers have high collection efficiency, operational problems associated with many wet scrubber installations reduce boiler steam output and, therefore, factory crushing rates. These problems include:

- Blockages that increase the pressure drop across the scrubbers. This reduces the gas flow through the boiler and, therefore, boiler steam output.
- Droplet carryover that causes build-up on and wear of boiler induced draft fans, reduces collection efficiency and generates complaints from the local community.

The need to improve the design of centrifugal wet scrubbers provided the motivation for this work which uses computational fluid dynamics (CFD) modelling and scale model experiments. The modelling and experiments were used to develop an improved understanding of the flow processes occurring in a centrifugal wet scrubber and to evaluate options for improving their performance. This project falls within the SRA key focus area of milling efficiency and technology.

A test rig with a scale model of a centrifugal wet scrubber module was constructed and used for velocity and pressure measurements and flow visualisation. The flow visualisation, with standard and high speed photography, and the measurements were used for CFD model validation and to gain a better understanding of the air and water flow patterns in the scrubber scale model. The flow visualisation identified the stages in the process water sheet break-up into droplets and the development of a swirling water bath that forms between the scrubbing and demisting vanes. The behaviour of this water bath affects droplet carryover and scrubbing efficiency. At very high air flows the water bath would extend beyond the demister vanes and lead to increased droplet carryover. At low air flow the water bath would not form and this would lead to reduce collection efficiency. Both these observations are consistent with the reported performance of full scale wet centrifugal scrubbers with high and low gas flows.

A site visit to two factories with centrifugal wet scrubbers was completed and the patterns of build-up in the scrubber modules were related to the predicted flow patterns in the scrubber scale model.

Observations from the scrubber scale model flow visualisation experiments were used in the development of a sub-model for dust particle capture by water droplets and water films. With this sub-model incorporated into the CFD code, the collection efficiency of a centrifugal wet scrubber can be predicted.

Observations from the scrubber scale model experiments and associated CFD modelling formed the basis of proposed design changes. Further modelling and modifications to the scrubber scale model were made to evaluate the proposed design changes. Feedback

was received from factory staff (end users) and the scrubber manufacturer on the proposed design changes and further modelling was carried out based on this feedback. This lead to updated scrubber design change recommendations.

Outputs from this project include the test rig and scrubber scale model, the improved modelling capability with the updated code, the CFD modelling and flow visualisation results and the recommended scrubber design changes.

Implementation of the recommended scrubber design changes should lead to reduced factory downtime, increased factory throughput, reduced emissions from sugar factory boilers and reduced complaints from the local community.

Some of the recommended design changes have already been implemented and it is likely that more widespread implementation of the design changes will occur.

Background

Wet scrubbers are the Australian sugar industry's most effective means of removing dust from boiler flue gas used in the Australian sugar industry. Currently wet scrubbers are used on more than 60% of the boilers in the industry but this proportion is increasing as new boilers with wet scrubbers installed replace older boilers that often use dry scrubbers. The more widespread use of wet scrubbers is significantly reducing the environmental footprint of the sugar industry.

In virtually all cases, the collection efficiency of wet scrubbers is high enough for boilers fitted with them to comfortably comply with the dust emission limits set by environmental authorities. However wet scrubbers often cause boiler operational problems such as excessive vibration, deposit build up on and wear of boiler induced draft fans due to water droplet carryover and reduced boiler steam output due to blockages in and/or around the scrubber. The profitability of a sugar milling operation is adversely affected by the reduced throughput and/or factory stoppages caused by boilers being taken off line for scrubber cleaning.

Wet scrubbers are effective and in many cases the most economically viable means of removing particulate and gaseous pollutants from gas streams (Calvert *et al.*, 1974; Lee *et al.*, 2013). They remove particles from a gas stream via the mechanisms of impaction, interception and diffusion (Schifftner and Hesketh, 1983). In most wet scrubbers, impaction, where the kinetic energy of the particle is used to penetrate the surface tension of the scrubbing liquid, is the dominant means of particle removal. Interception occurs when particles meet the droplets at angles much less than 90° and at lower relative velocity than that required for impaction. Diffusion, where particles migrate and come into contact with droplets due to gas density and turbulence fluctuations, is only important for very small particles (< 5 μm).

Several designs of wet scrubbers are used to control emissions from sugar factory boilers around the world. The most common designs include venturi scrubbers (fixed or variable throat) (Jones, 1949), sieve plate (flooded and impingement) scrubbers, fixed vane centrifugal scrubbers, spray tower scrubbers and static bath impingement scrubbers (Lora and Jativa, 1999; Moor, 2007; Pennington, 1999). In the Australian sugar industry only the fixed vane Ducon and Clyde Carruthers style and the multi-venturi design wet scrubbers are currently used as the primary means of dust collection. Sketches of these scrubber designs are shown in figures 1, 2, and 3 respectively. Most of the wet scrubbers in the Australian sugar industry are the fixed vane Ducon style (figure 1).

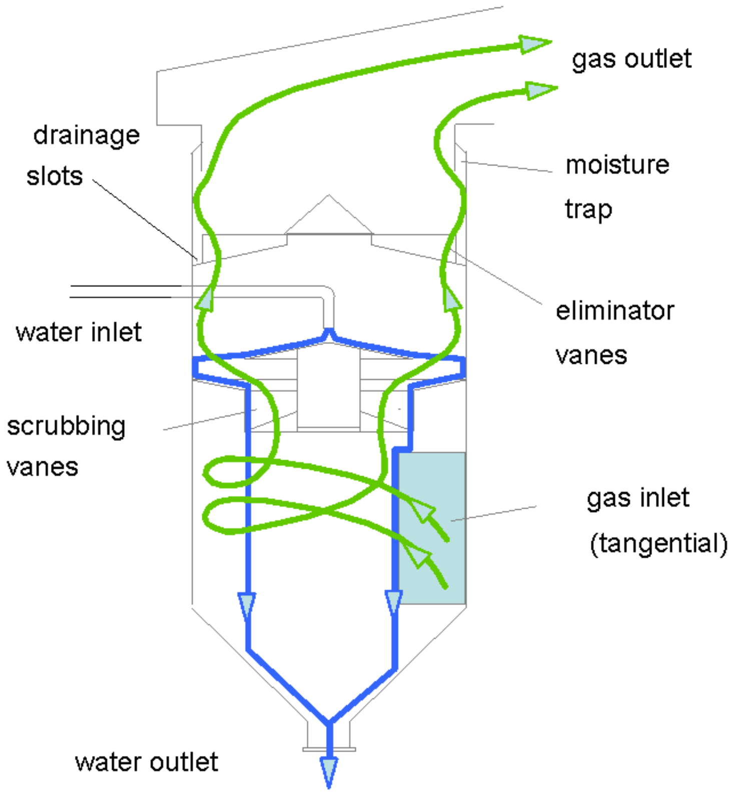


Figure 1 **Ducon style fixed vane scrubber.**

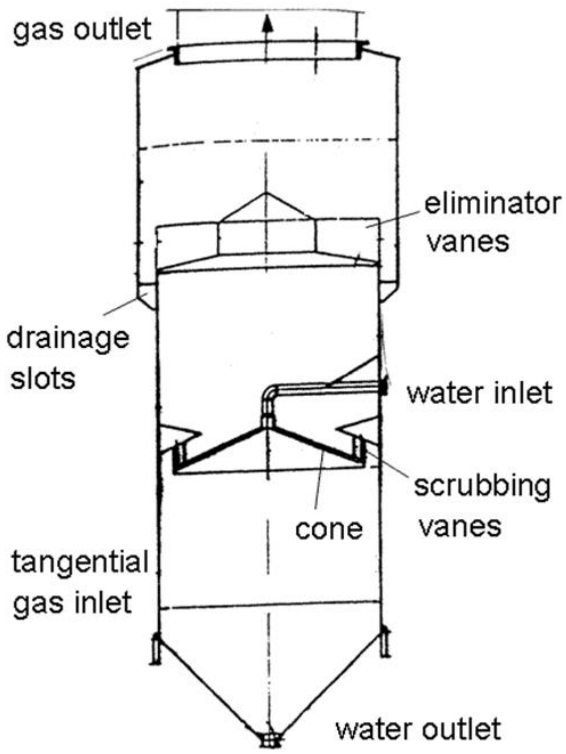


Figure 2 **Clyde Carruthers style fixed vane scrubber.**

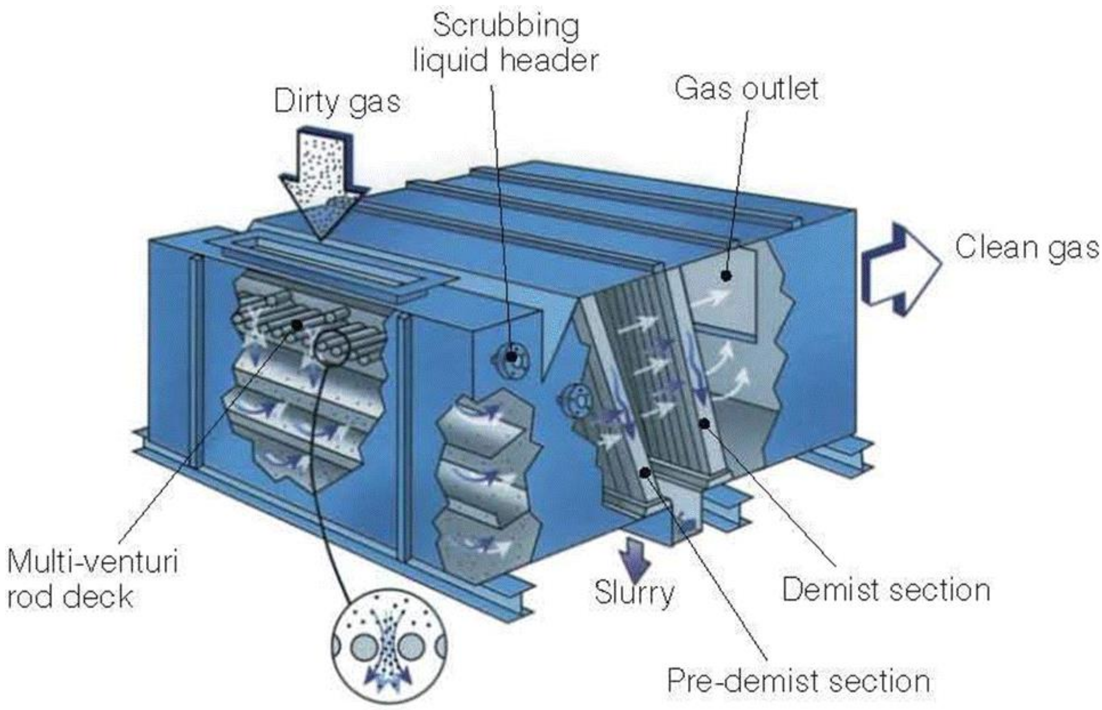


Figure 3 **Multi-venturi fixed vane scrubber (www.mikropul.com – accessed on the 6-6-2008).**

The design of wet scrubbers has traditionally been based on past experience and rules of thumb. In some cases CFD has been used in the design process but this modelling usually only considers the gas flow patterns only. Modelling carried out by QUT in the last few years is more sophisticated and includes the motion of the water droplets and evaporation, but more development work is required to account for effects such as the flow of bulk water, the agglomeration of water droplets into bulk water and the interaction between dust particles and bulk water or water droplets.

The flows in a wet scrubber are very complex due to the interaction between dust particles, water droplets, bulk water and flue gas. The flue gas flow is turbulent and there are multiple phases (gas, liquid and solid).

CFD codes that predict gas flow patterns are based on conservation equations for mass, momentum and energy (Bird *et al.*, 1960; Roache, 1976). Most of the major CFD codes in use are based on the TEACH set of codes developed at Imperial College (Patankar and Spalding, 1972).

The modelling becomes significantly more complex when more than one phase is present. Two phase modelling, which considers a gas phase and a dispersed phase (usually liquid or solid) has been the focus of much modelling effort (Drew, 1983; Hinze, 1972; Melville and Bray, 1979). Most techniques used for solving two phase flow problems can be categorised as either Eulerian or Lagrangian.

An Eulerian approach (Harlow and Amsden, 1975; Whitaker, 1973), often called a 'two fluid' model, treats the cloud of particles as a continuum. Conservation equations can be formulated for the particle phase in the same manner as for the gas phase. These conservation equations will include the volume concentrations (void fractions) of each phase. To do this an effective diffusion coefficient for the particle phase must be assumed. This has been an area of much study. Coupling between the two phases is modelled by source terms in the respective conservation equations. A significant advantage of Eulerian methods is that numerical techniques developed for solving the gas phase flow can be applied to the particle phase. However, storage requirements for Eulerian schemes can become quite large when a number of representative particle size classes are used because each size class usually requires a separate conservation equation. As gas phase numerical techniques are used for the particle phase, the solution for the particle phase will be affected by numerical instabilities and numerical diffusion. In addition, modifications to the particle effective diffusion coefficient must be made at the wall and the particle slip velocity at the wall must be established.

A Lagrangian approach involves following the motion of a representative number of particles by solving an equation of motion for each particle. It was proposed (Migdal and Agosta, 1967) that the particles could be regarded as a source of mass, energy and momentum to the continuum phase (two way coupling). This was later formalised by developing the PSI-Cell (Particle-Source-In Cell) method (Crowe *et al.*, 1977) that has since become widely used. This approach ignores particle to particle interactions and thus cannot be used for very high concentrations of the particle phase. Simulating the effect of turbulent dispersion can be a problem with this method. One option used a diffusion 'force' (Dukowicz, 1980) requiring the selection of a diffusion coefficient. Others (Leonard *et al.*, 1980) have also treated turbulent dispersion as a Fickian diffusion process while modelling the flow in an electrostatic precipitator. Another method for modelling turbulent dispersion is the Monte Carlo technique (Gosman and Ioannides, 1981). The turbulent gas flow field is represented as steady flow plus a random velocity fluctuation. Many particles must be used to simulate the cloud, thereby increasing simulation run time. Consequently, this method can be computationally expensive but has the advantage of low storage requirements.

A Lagrangian model of the particle phase allows easier treatment of particle wall interactions and gives more information on particle trajectories and residence times. Furthermore, reasonable predictions of particle trajectories are possible even when a rather coarse flow grid is used. Lagrangian techniques have been used to model the flow in a cyclone separator (Crowe and Pratt, 1974) and the flow of coal particles in an axisymmetric furnace (Lockwood *et al.*, 1980). This approach has also been used to model spray units (Palaszewski *et al.*, 1981).

The relative merits of Eulerian and Lagrangian approaches have been investigated by many workers (Adeniji-Fashola and Chen, 1990; Crowe, 1982; Durst *et al.*, 1984). In one case (Durst *et al.*, 1984) both approaches to model two phase vertical pipe flow were compared. They found that Lagrangian particle tracking was superior when particle accelerations were high. An Eulerian approach was better suited to cases with high particle concentrations.

Earlier modelling of a Ducon style wet scrubber carried out at the Sugar Research Institute (Downing *et al.*, 2000) considered only the gas flow patterns because the commercial CFD modelling software available at the time was not able to represent a dispersed phase (dust particles, water droplets) on the mesh required for the model. The FURNACE code (Boyd, 1986; Dixon *et al.*, 2005; Luo and Stanmore, 1994; Mann, 1996; Woodfield, 2001) is more sophisticated and includes the motion of the water droplets (with two way coupling between the gas and particle phases) and evaporation. However this code, while ideally suited for modelling the flow patterns and combustion in bagasse and coal fired boilers, can only solve on a cartesian structured grid which makes it difficult to model the internal components of a scrubber. Furthermore, the k-ε turbulence model used by the FURNACE code is known to be deficient in strongly swirling flows (Weber *et al.*, 1990).

CFD has been used by several researchers to predict single and two-phase flow in dry gas cyclones (Boysan *et al.*, 1982; Chuah *et al.*, 2006; Cortes and Gil, 2007; Griffiths and Boysan, 1996; Karagoz and Kaya, 2007) and multi-phase flow in a dense medium cyclone (Narasimha *et al.*, 2007). The earlier work on single phase cyclone flow identified that the k-ε turbulence model could not accurately predict the flow patterns. More recent CFD modelling of a sampling cyclone achieved reasonable agreement between predicted and measured axial and tangential velocities when using an algebraic turbulence model for the gas phase (Chuah *et al.*, 2006). This modelling work only had one way coupling between the gas and particle phases.

Outlet droplet distributions for wet venturi scrubbers have been predicted with some success using a two-dimensional CFD model (Ahmadvand and Talaie, 2010). More complex CFD models of a venturi scrubber that take into account two dispersed phases (droplet and dust particles) have also been developed (Pak and Chang, 2006). Lagrangian particle tracking is used to predict the motion of droplets and dust particles. This model takes into account atomisation of the liquid jet (Wu *et al.*, 1997) and capture of the dust particles by the droplets (Mohebbi *et al.*, 2003).

Another wet venturi scrubber investigation included the collection efficiency tests by Costa *et al.* (2005) who operated a test scale venturi scrubber with mineral salt as the dispersed phase. Collection efficiencies were measured for different scrubbing liquid flow rates and air velocities. Experimental results and CFD predictions were compared by Guerra *et al.* (2012); this work showed how scrubber pressure drop is affected by the number of liquid injection orifices. Ali *et al.* (2015) have measured the collection efficiency of a venturi scrubber using a filtration technique and compared the results with a previously developed expression for collection efficiency via inertial impaction (Calvert, 1970), which is usually the most important mechanism for dust collection in wet scrubbers (Pak and Chang, 2006).

The studies noted above are just some of the experimental and CFD investigations of dry cyclones and wet venturi scrubbers. By contrast there are few published studies of wet centrifugal scrubbers like the Ducon and Clyde Carruthers used in the Australian sugar industry. This could be because there are few empirical relations, experimental data and measurements that can form the basis of such an investigation like there are for dry cyclones and wet venturi scrubbers.

The work carried out in this project, which combines experimental and CFD investigation into centrifugal wet scrubbers, therefore fills a void in the published literature.

Project Objectives

The objectives of the project are to gain an improved understanding of the flow processes in wet scrubbers. These flow processes include but are not limited to:

- Flue gas and droplet flows;
- Agglomeration of droplets;
- Flow of bulk water; and
- Interaction between dust particles and water droplets.

With the improved modelling capability, existing scrubber designs can be simulated with the aim of determining design modifications for improved scrubber performance.

Methodology

Construction of the test rig

After the initial review of relevant literature and enrolment of the PhD student, a laboratory scale model of a wet scrubber was constructed. This model, along with the extraction fan and connecting duct work, form the test rig that was used for flow visualisation CFD model validation. The test rig, in conjunction with CFD modelling, was used to evaluate scrubber design modifications. The test rig is located at the QUT pilot plant facility at Banyo and is shown in figure 4. A close up view of the scrubber scale model is shown in figure 5.

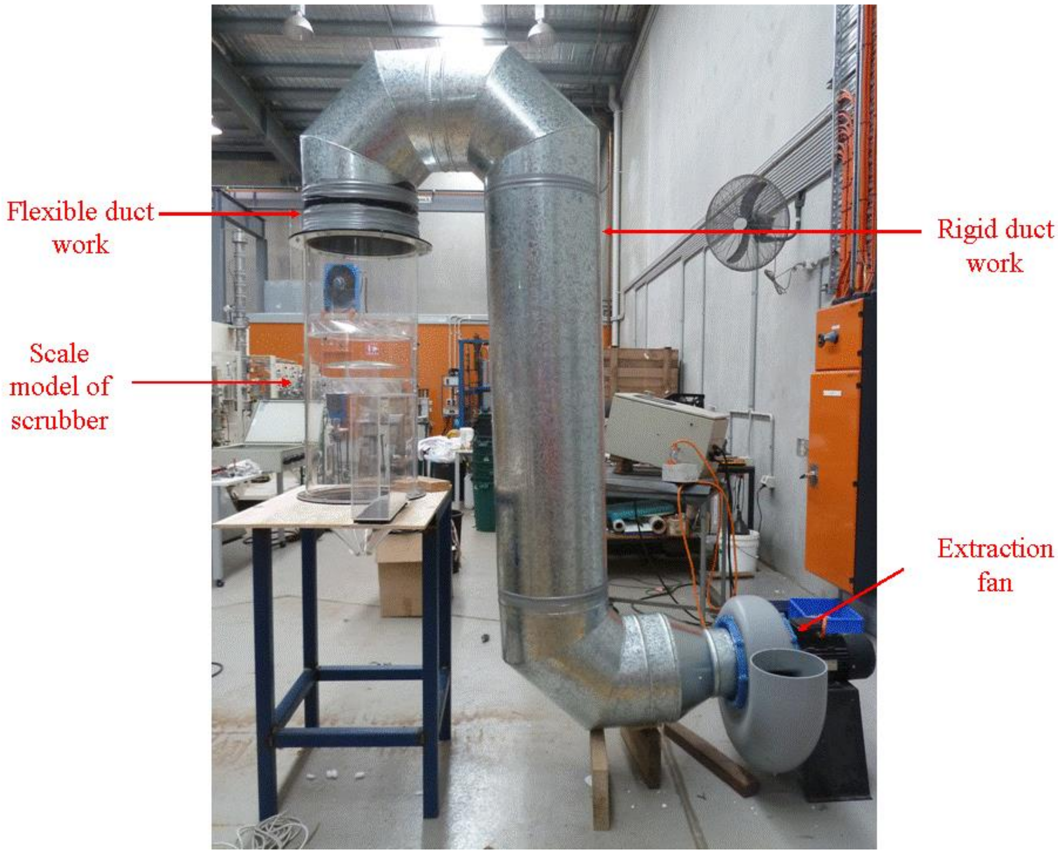


Figure 4 Test rig for wet scrubber scale model.

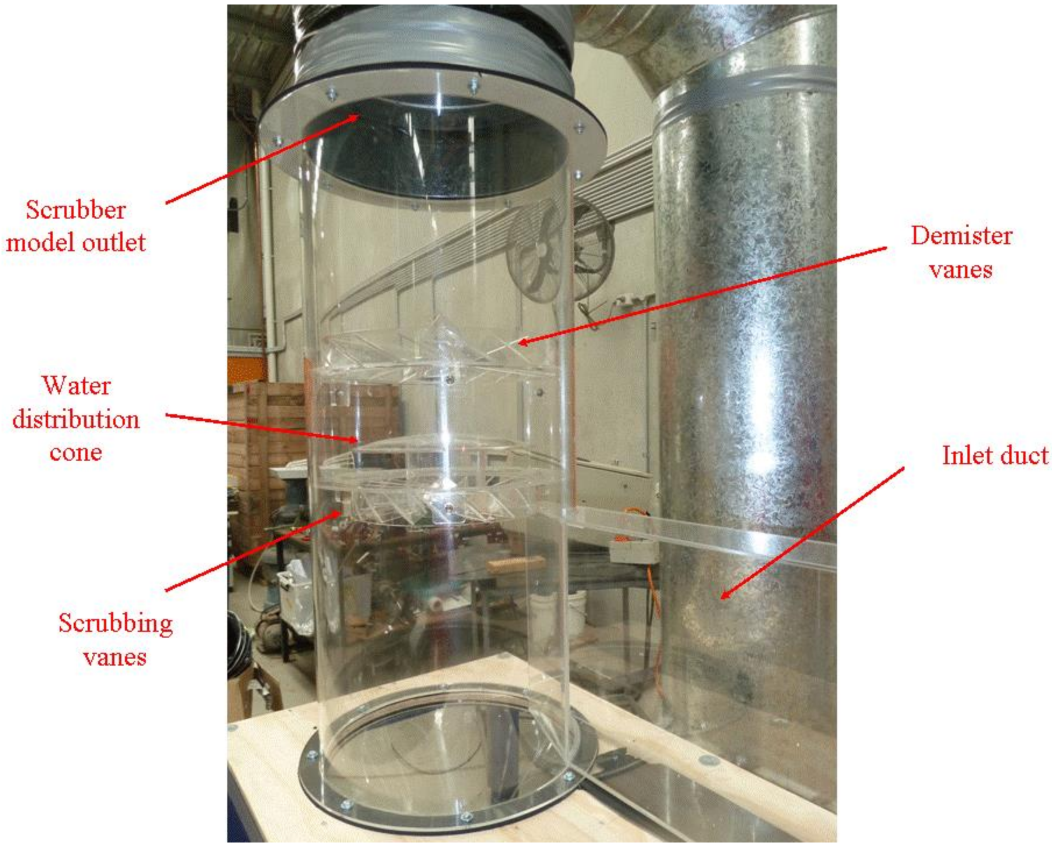


Figure 5 **Wet scrubber scale model.**

The scale model is made of transparent acrylic. Flexible and rigid duct work is used to connect the scale model to the extraction fan. The small length of flexible duct work was used to reduce the amount of fan vibration transmitted to the scale model. It was originally proposed that the fan would be connected to the inlet duct, rather than the outlet of the scale model. However the as constructed arrangement is superior because the velocity and pressure measurements are not be affected by fan generated turbulence. The as constructed arrangement is also closer to the actual arrangement of a wet scrubber module in a boiler (upstream of the induced draft fan). The extraction fan can run at full speed without affecting the structural integrity of the scale model or duct work.

The scrubber scale model is approximately one tenth the size of a full scale scrubber used to remove dust from boiler flue gas. This meant that to achieve full Reynolds number similarity with the full scale scrubber the air velocities through the SSM needed to be higher than 100 m/s, which was clearly impractical. Therefore, in the experiments, the air flows through the scrubber scale model were adjusted using the variable speed controller so that the air velocities in the scrubber scale model would approximately match the gas velocities in the full scale scrubber. This gave Reynolds numbers based on the air velocities and SSM diameter greater than 100 000 which corresponds to fully turbulent flow. Velocity and pressure measurements were made using a multi-meter equipped with a hot wire anemometer and a differential pressure meter.

The test rig was operated with different air flows without and with water addition. To avoid damaging the hot wire anemometer, velocity measurements were only taken when the rig was operated with air flow only. On some of the occasions when the rig was operated with air flow and water addition, high speed cameras (X-Stream XS-4 and HiSpec 1) were used to analyse the water flow through the scrubber scale model.

Upgrade of the CFD model

ANSYS Fluent is arguably the most powerful CFD package for simulating multi-phase flows and was chosen for the modelling work in this project. One of the strengths of the code is its ability to model the dispersed phases using Lagrangian particle tracking or by solving a separate conservation equation for each of the dispersed phases (the Eulerian or volume of fluid approach). However the code, while powerful, cannot represent all flow phenomena occurring in the wide range of applications that the code is used to simulate. For this reason the code was written to allow the user the option of overriding or adding to many of the sub models used. This is done with user defined functions.

A review of the relevant literature was carried out to determine the most appropriate approaches for simulating the various flow processes occurring in a wet scrubber. In some cases the preferred approach is already available in ANSYS Fluent while for others, user defined functions are required to better simulate the multiphase flow in a wet scrubber.
One of the important processes occurring in tangential entry centrifugal wet scrubber is the flow of bulk water, specifically over the water distribution cone and down the inside wall of the scrubber module. This can already be simulated using the Eulerian approach available in ANSYS Fluent.

Effective break-up of the water curtains that form over the edge of the water distribution cone and underneath the scrubbing vanes is essential for good scrubbing efficiency and for the reduction of water droplet carryover. The thickness of the water curtain can be estimated by carrying out Eulerian modelling of the water flow over the water distribution cone and the flow down the outside of the scrubbing vanes. The predicted water curtain thickness was used as an estimate of the droplet mean particle size and an assumed initial particle size distribution can be used to determine droplet sizes for Lagrangian particle tracking in a manner similar to earlier modelling work on venturi scrubbers (Pak and Chang, 2006).

The population balance model (Hounslow *et al.*, 1988) has been used for bubble columns, aerosols and atomization of liquid jets. It can predict the break-up of larger droplets into smaller droplets and the coalescence of smaller droplets into larger droplets. The population balance model is available as an option in the CFD code ANSYS Fluent.
The break-up of droplets is strongly dependent on the Weber number (We) which is defined for a droplet in a gas flow as (Pilch and Erdman, 1987):

$$We = \frac{\rho V^2 d}{\sigma}$$

where ρ is the density of the gas, V is the droplet velocity relative to the gas, d is the droplet diameter and σ is the surface tension of the droplet material. The break-up mechanism is strongly dependent on the Weber number as shown in figure 6.

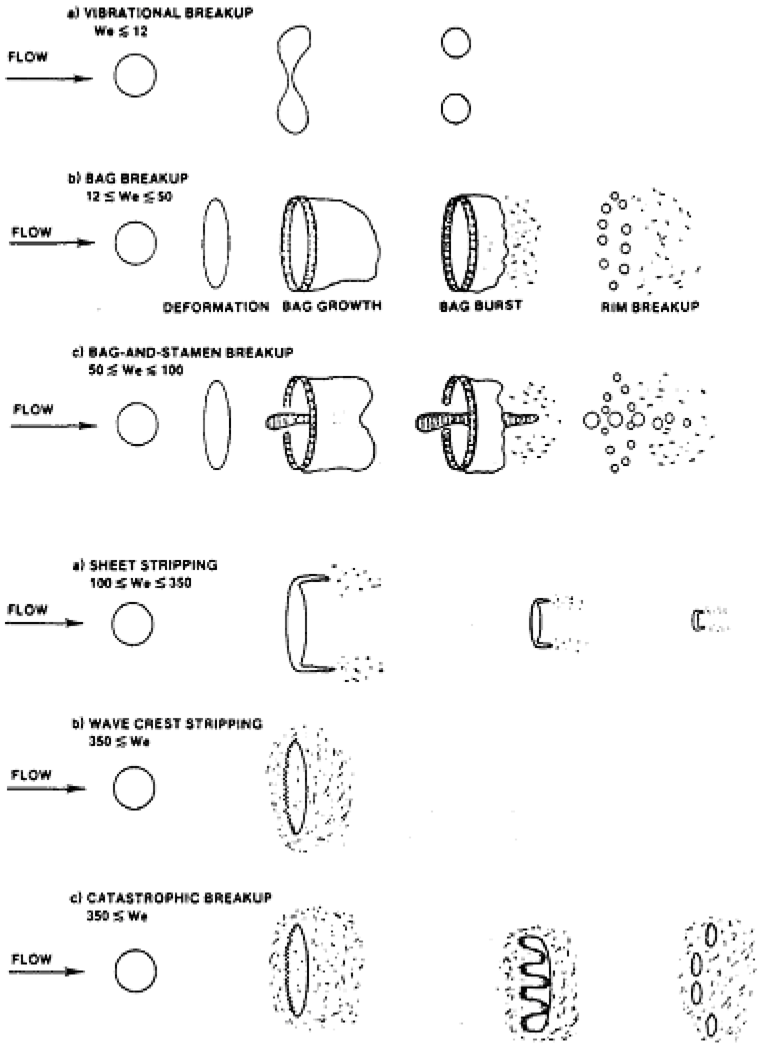


Figure 6 Droplet break-up mechanisms (Pilch and Erdman, 1987).

Droplet collisions are modelled in ANSYS Fluent with the collision volume approach (O'Rourke, 1981) which predicts the probability that two particles in the same computational cell will collide.

In this project the above approach for droplet coalescence has been updated using the macro DEFINE_DPM_SPRAY_COLLIDE to include a kinetic energy test. This kinetic energy test takes into account that a collision between a dust particle and a droplet will only result in droplet capture if the kinetic energy associated with the collision exceeds the work required for droplet capture defined by Pemberton (1960). The criterion for dust particle capture by a droplet therefore can be expressed as:

$$\frac{1}{2} m_{dust} \left|\tilde{V}_{rel}\right|^2 > \frac{8\pi r^2 \sigma}{3}$$

where m_{dust} is the mass of the dust particle and $\tilde{V}_{rel}$ is the velocity of the dust particle relative to the droplet. Note the above assumes that the dust particle is smaller than the water droplet. The assumption is justified because preliminary calculations have shown that the minimum stable droplet diameter for the gas velocities encountered in a wet scrubber is of the order of 1 mm while nearly all dust particles from a bagasse fired boiler have diameters less than 0.1 mm.

The above kinetic energy approach was also used to model the capture of dust particles by the liquid film on the inside wall of a centrifugal wet scrubber.

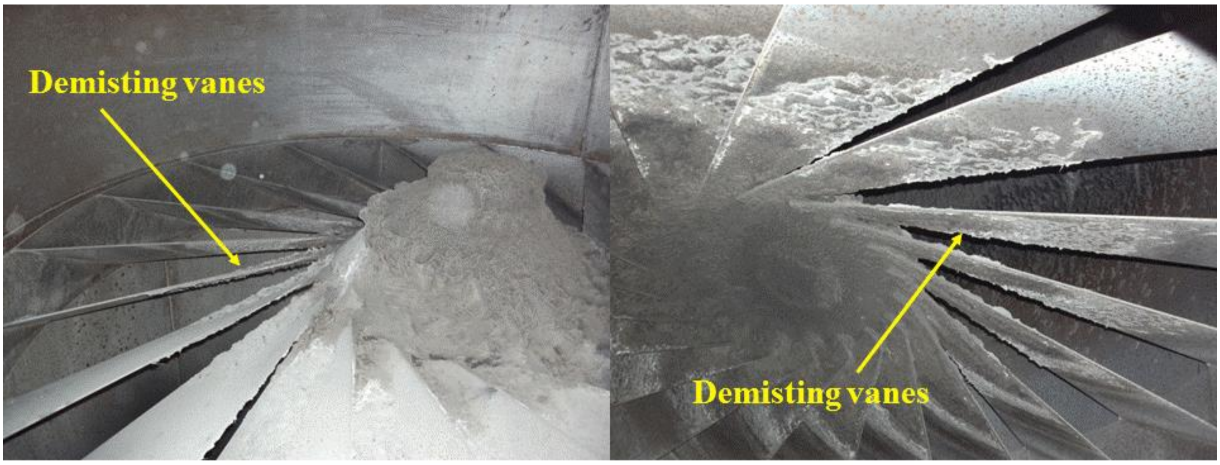
In a centrifugal wet scrubber virtually all the droplet coalescence would be expected to occur on or near the inside wall. Droplets that do not coalesce onto the wall are likely to be carried over from the scrubber and cause build-up of ash on the induced draft fan and black rain emissions from the stack. It has been found that particles above a certain size are likely to coalesce onto a surface while smaller particles are unable to remove the air between the contact regions of the wall film and the droplet. This critical size depends on a modified version of the Weber number based on the droplet density and collision velocity (Pan and Law, 2007). This critical collision Weber number is used as a criteria for the droplet being collected at the scrubber walls and is introduced as a user defined function in ANSYS Fluent using the Macro DEFINE_DPM_BC. However it has also been found that if this collision Weber number is too high the droplet hitting the wall may fragment into smaller droplets (Orme, 1997). These smaller droplets could have collision Weber numbers too small for coalescence with the film on the wall which means they will bounce off the wall and possibly exit the scrubber. This is also incorporated into the user defined function for wall collisions.

The collection of dust particles by free stream droplets and the wall film were taken into account using the approach described above. It is noted that the droplets that are not collected by the wall film exit the scrubber with the gas flow as carryover. The dust particles collected by these droplets are not in fact removed from the gas flow. The code modifications take into account the reduction in dust collection efficiency due to droplet carryover.

Wet scrubber inspections

Inspections of wet scrubber modules were carried out at Bingera Mill and Isis Mill. The inspections included the inside of scrubber inlet duct, underneath the scrubbing vanes, in between the scrubbing vanes and demisting (eliminator) vanes and above the demisting vanes. Figure 7(a) shows the build-up of ash below the demisting vanes of a scrubber module and figure 7(b) shows the build-up above the demisting vanes. Little build-up was seen around the scrubbing vanes and pressure disc as seen in figure 8(a) but figure 8(b) shows significant build-up on the roof of the scrubber inlet duct.

The scrubber inspections identified design variations such as the convergence of the scrubber gas inlet duct, the use of a pressure disc below the scrubbing vanes and a dam on the water distribution cone. Figure 9(a) shows a water distribution cone without a dam and figure 9(b) shows a water distribution cone with a dam.



(a) (b)

Figure 7 **Build-up (a) below and (b) above the demisting vanes.**

(a) (b)

Figure 8 **(a) Scrubbing vanes and pressure disc and (b) deposit on top of scrubber inlet duct.**

(a) (b)

Figure 9 **Water distribution cone (a) without and (b) with a dam.**

Results

Test rig scale model measurements and observations

Velocity and pressure measurements were carried on the test rig scale model for different air volumetric flow rates. The air volumetric flow rates through the scale model were chosen so that average air velocities in the scale model would be similar to the average gas velocities in a full scale scrubber at different boiler steam loads. The measured velocity magnitudes have been used for CFD model validation.

Figures 10 and 11 show measured tangential and axial air velocities in a plane below the scrubbing vanes for traverses parallel to and perpendicular to the direction of the flow in the inlet duct with an air volumetric flow rate of 0.2 m^3/s. The pressure drop across the scale model corresponding to these measurements was approximately 380 Pa. Figure 10 shows that the velocity distribution is not symmetrical with higher velocities on the side of the scale model opposite to the direction of flow in the inlet duct (bottom of the sketch in figure 10). The minimum velocities are also closer to the side of the scale model opposite the direction of flow in the inlet duct. Velocity measurements along the traverse perpendicular to the direction of flow in the inlet duct also show non symmetrical flow with highest velocities near the inside wall of the scale model (left of the sketch in figure 11) and near the protrusion of the inlet duct into the scale model.

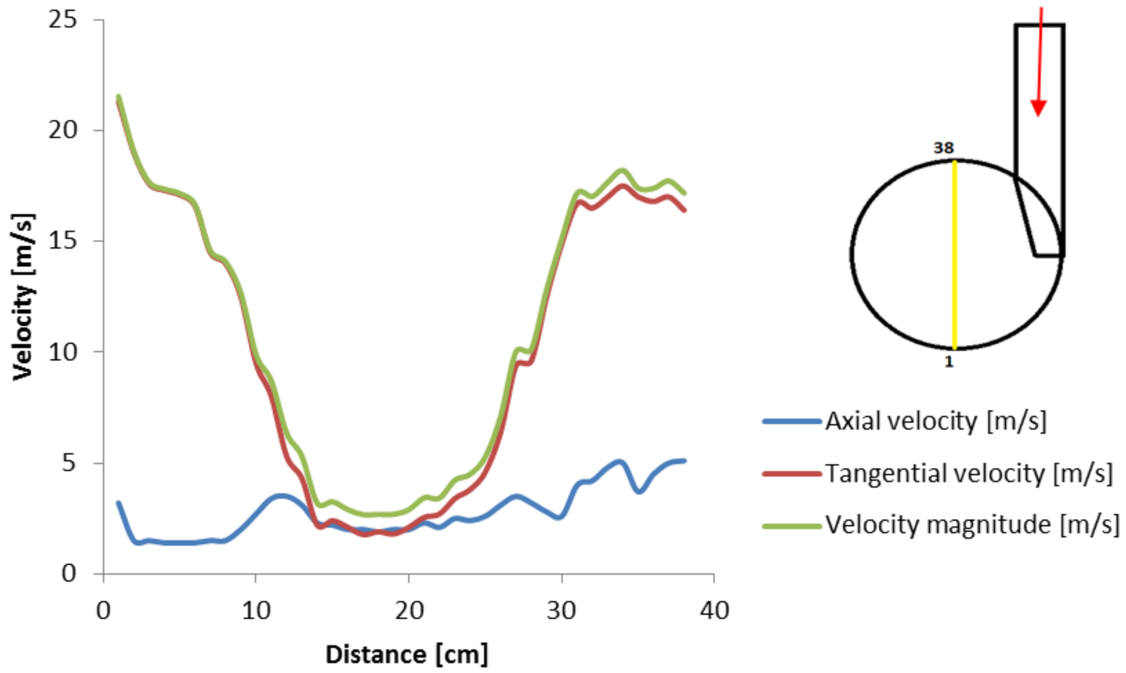


Figure 10 Measured velocity components and velocity magnitudes (m/s) along a traverse parallel to the longitudinal axis of the scale model inlet duct. Locations of the 1 and 38 cm distances are shown on the sketch.

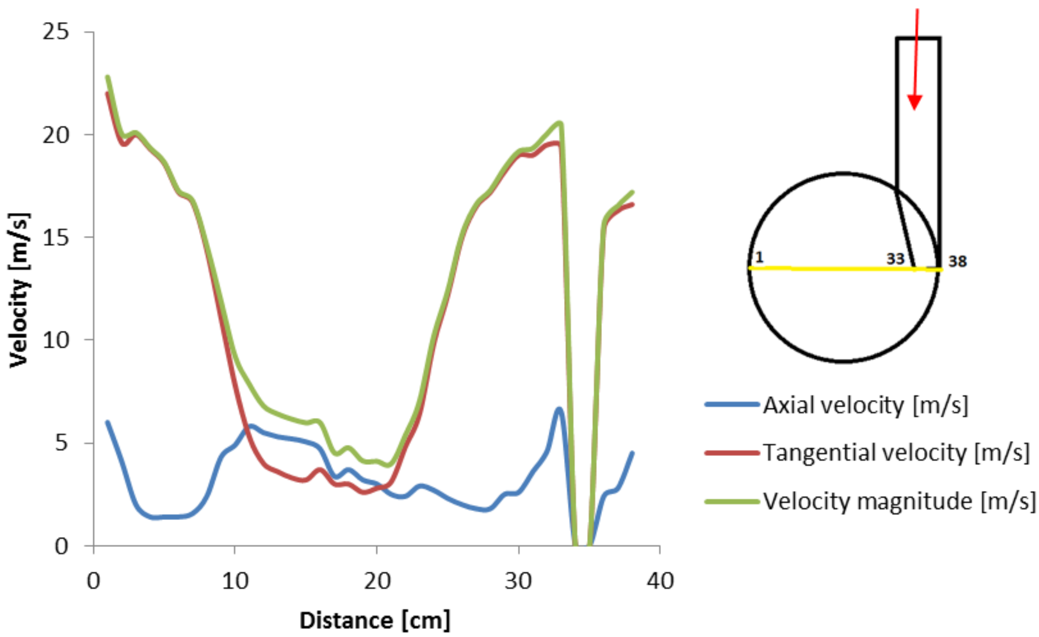


Figure 11 Measured velocity components and velocity magnitudes (m/s) along a traverse perpendicular to the longitudinal axis of the scale model inlet duct. Locations of the 1, 33 and 38 cm distances are shown on the sketch.

Figure 12 shows the measured velocity magnitudes between the scale model scrubbing vanes. The velocity distribution is non uniform with velocity magnitudes varying by a factor of 2.9.

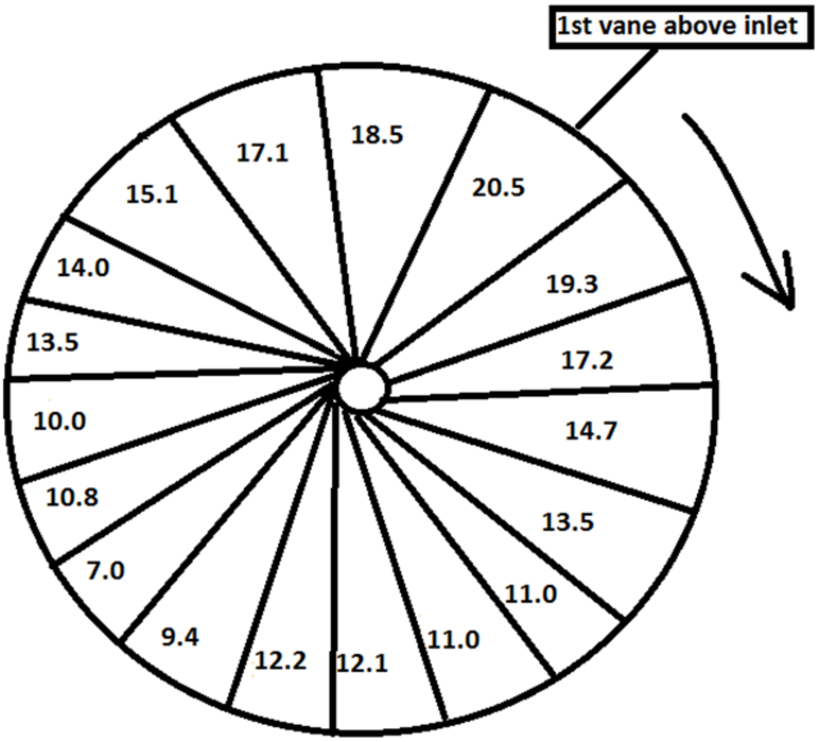


Figure 12 Measured velocity magnitudes (m/s) between the scale model scrubbing vanes.

Further tests were carried out with water added to the trough on the water distribution cone as air flowed through the scale model. The rate of water addition was chosen so that the rate of water addition per unit inlet air volume flow would be approximately equal to the design rate of water addition per unit inlet gas volume flow for a full scale scrubber module. It was not possible to measure the air flow distribution in the main body of the scale model because the water would have damaged the hot wire anemometer but it was still possible to measure velocities in the scale model inlet duct (and therefore the air volumetric flow rate) and the pressure drop across the scale model. Tests for different air flow rates found that water addition reduced the volumetric flow through the scale model by approximately 7% and increased the pressure drop by approximately 23%. This corresponds to an increase in fan power consumption of approximately 14%. This increase in pressure drop with water addition is much less than that reported in the literature for venturi scrubbers (Schifftner and Hesketh, 1983). Note that in a full scale operating scrubber there will be a reduction in gas volume flow as the gas is cooled by the water; this should partially compensate for the increase in pressure drop due to water addition. This effect is not picked up in the scale model experiments because air enters at ambient temperature and there will be little cooling of this air when water is added.

Figure 13 shows that a highly agitated water bath forms between the scrubbing and demisting vanes with water addition. It appears that a lot of the gas cleaning can occur as it passes through this water bath. It was also noted that at high rates of water addition this water bath overflowed past the demisting vanes. This could be a mechanism for the water droplet carryover that often occurs when too much water is added to an operating tangential entry fixed vane wet scrubber. Furthermore, it was found that when the air flow rate was very high, the water bath lifted up into the demisting vanes. This is a likely mechanism for the increased droplet carryover experienced by tangential entry fixed vane scrubbers when a boiler is operated with very high gas flows resulting from tramp air leakage and/or the boiler operating at high steam loads.

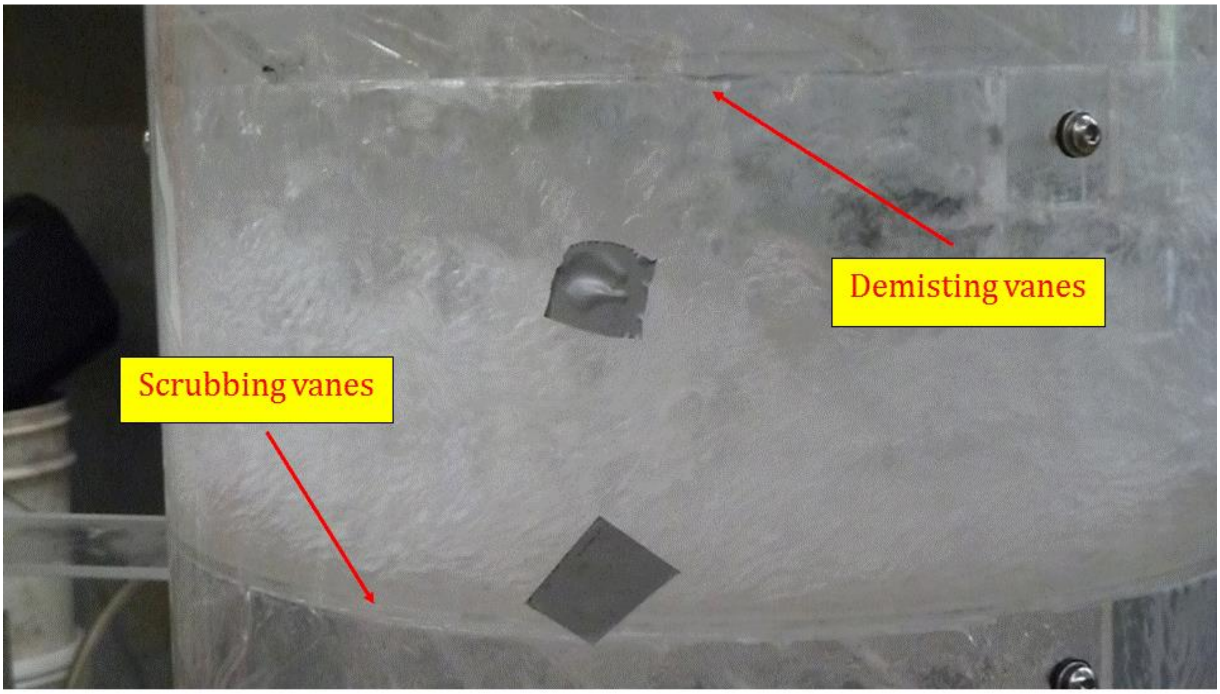


Figure 13 **Part view of the scrubber scale model showing the agitated water bath between the scrubbing and demisting vanes.**

Flow visualisation in the scrubber scale model

Images from the flow visualisation experiments are shown in figure 14. Figure 14 (a) shows a standard camera image of the flows through the scrubber scale model. Note that with this image the wall water film obscures the internal flow patterns. The image from the high speed camera taken at 250 frames per second (fps) in figure 14 (b) clearly shows the agitated water occupying most of the volume between the scrubbing and demisting vanes. A close up image of some droplets above a demister vane taken with a very small exposure time (5000 fps camera speed) is shown in figure 14 (c) and figure 14 (d) shows some droplets in the above demister vane region taken with the high speed camera operating at 1000 fps. A close up view showing the breakup of a water sheet from the water distribution cone is shown in figure 14 (e). Figure 14 (f) shows a close up view between two scrubbing vanes taken by the high speed camera at 2000 fps. The liquid film flowing down the top of the vane shown in the bottom of the image is broken up into ligaments as it flows over the bottom of the vane by the upward air flow.

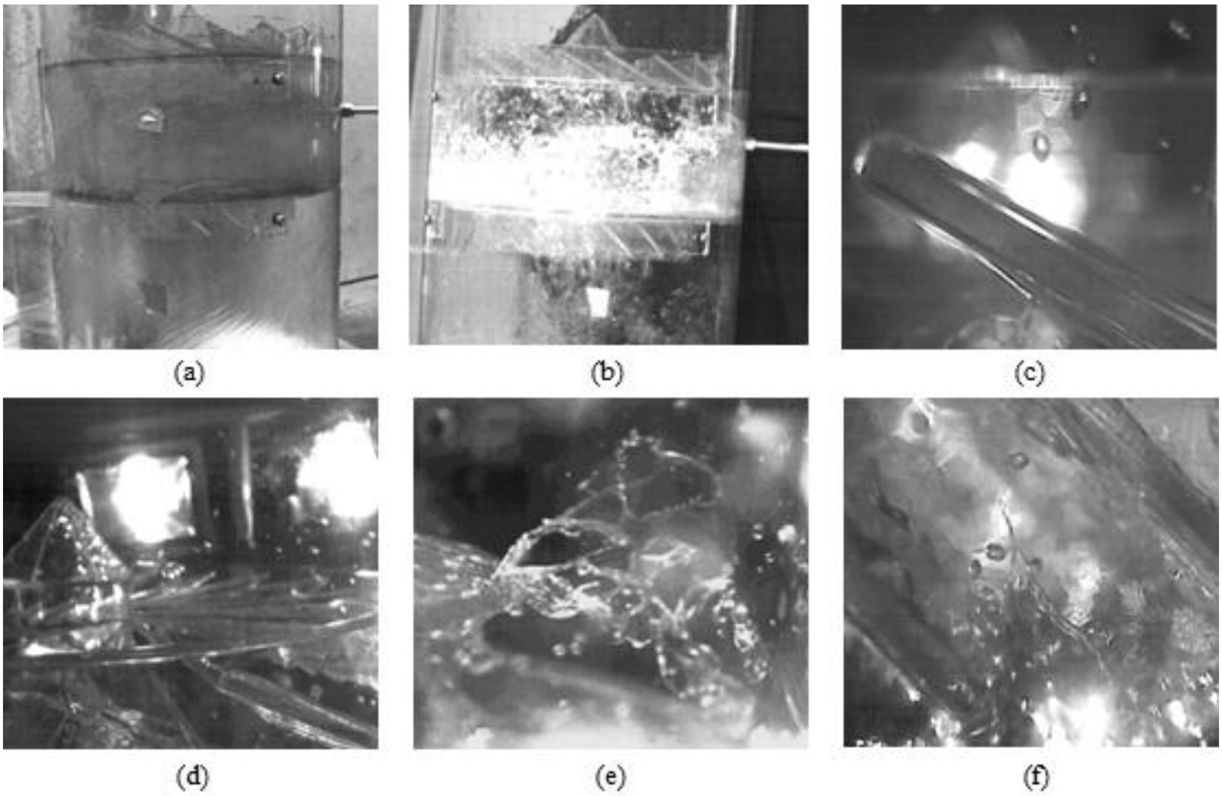


Figure 14 Scrubber scale model images from the flow visualisation experiments: (a) regular camera, (b) high speed camera at 250 fps, (c) close up view showing part of a demister vane at 5000 fps, (d) close up view showing region above demister vanes at 1000 fps, (e) close up view showing the breakup of a sheet from the water distribution cone at 1000 fps, (f) close up view showing the liquid film separating from the scrubbing vanes breaking up into ligaments and droplets at 2000 fps.

Test rig scale model air flow modelling

Figure 15 shows a side elevation view of the predicted air velocity vectors in the scrubber scale model. The green colour and longer vector lengths in the figure correspond to higher air velocities. The highest air velocities occur near the wall of the scrubber scale model below the scrubbing vanes and through the scrubbing vanes. Recirculation zones are predicted to occur around the centreline of the scrubber scale model below the scrubbing vanes and demisting vanes and above the demisting vanes. The predicted recirculation zones above and below the scrubbing vanes correspond to the observed ash build-up in existing scrubbers seen in figure 7. The high predicted velocities through the scrubbing vanes seen in figure 15 correspond to the minimal ash build-up on the scrubbing vanes seen in figure 8(a).

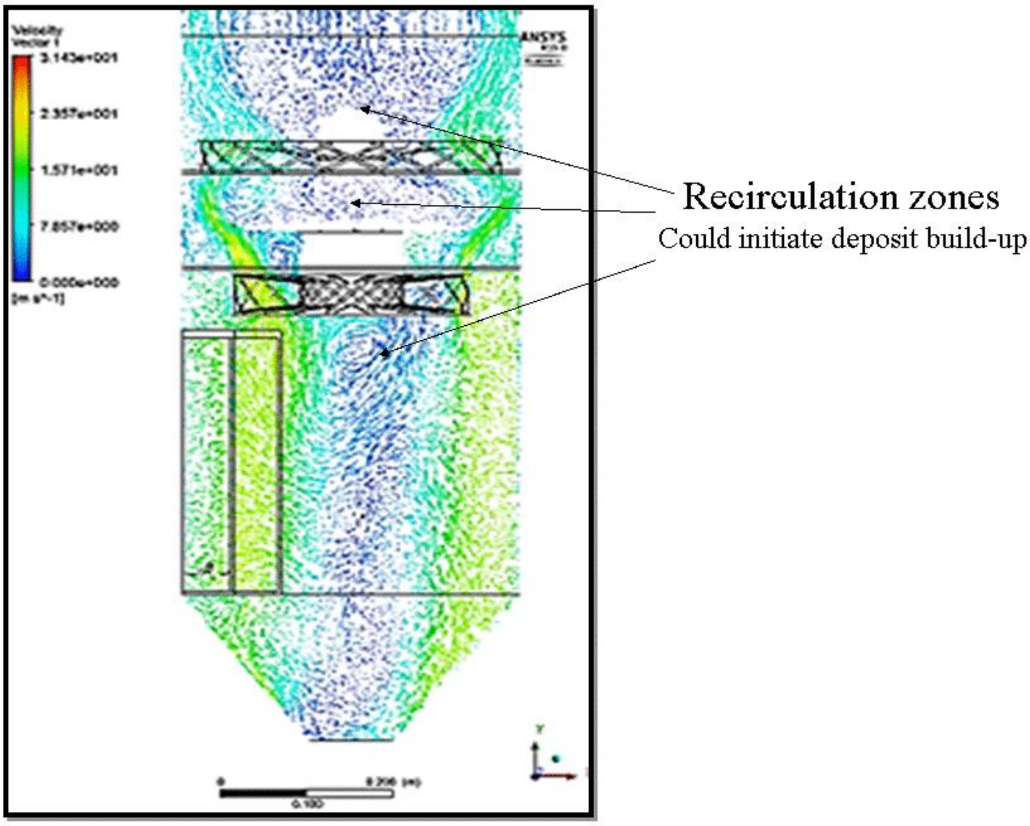


Figure 15 **Predicted air velocity (m.s^{-1}) distribution through the scrubber scale model.**

Initial full scale scrubber gas flow modelling

CFD simulations of two different full scale scrubber modules (scrubber module A and scrubber module B) were performed. These modules are different size and have similar but not identical geometries. These simulations used calculated gas flows based on operating data for the two boilers.

The first simulations carried out only considered the gas flow distributions through the scrubber modules without any water addition and it was assumed that the properties of the gas were the same as those of air at 25 °C (test rig scale model conditions) and that these properties did not change. This is clearly not the case in an actual scrubber but the purpose of the first set of simulations was to predict full scale scrubber module gas velocity distributions that could be directly related to the test rig measurements and the CFD simulations of the test rig scale model.

Figures 16, 17 and 18 show plan views of the predicted gas velocity distribution at two different elevations in scrubber module A for a modelled gas flow rate of 34 kg/s. The elevation of the plane of view in figure 16 is through the scrubbing vanes, the elevation of the plane of view in figure 17 is above the water distribution cone and the elevation of the plane of view of figure 18 is through the demisting vanes. The scrubber module gas inlet duct is to the upper right and below the planes of view of all figures. Figure 17 shows the predicted gas velocities are higher on the gas inlet duct side of the scrubbing vanes and the lower gas velocities are on the opposite side of the scrubber module.

Above the water distribution cone the flow is a lot more concentrated towards the inside wall and the location of the highest gas velocities has moved clockwise around the scrubber module. The flow through the demister vanes (figure 18) becomes a lot less concentrated towards the inside walls of the scrubber module and the predicted gas velocities are lower. As the flow spirals up through the demister vanes the location of the highest gas velocities has moved clockwise to the opposite side of the module to the gas inlet duct. All three plots show highly non-symmetrical flow.

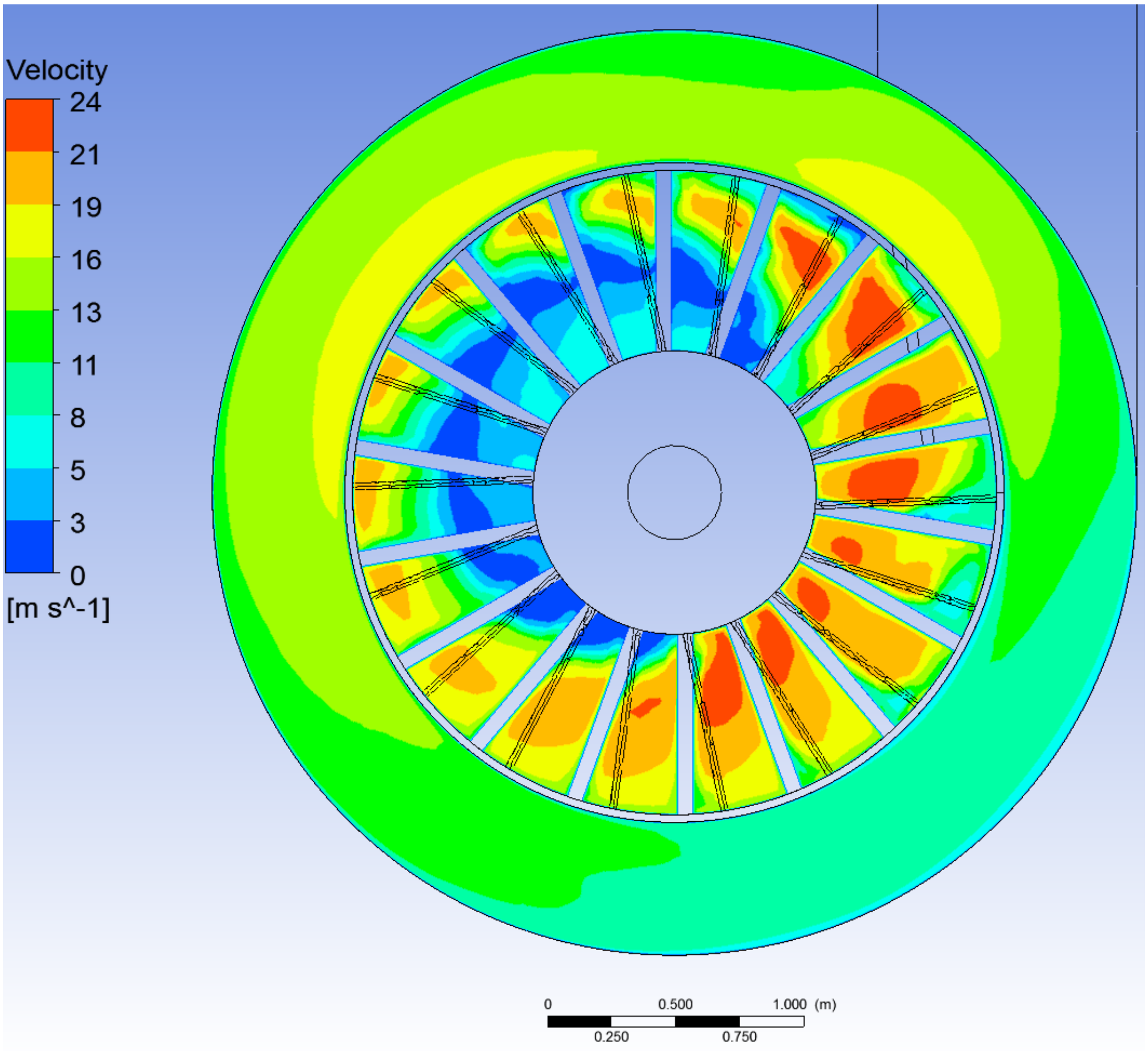


Figure 16 Plan view of the predicted gas velocity (m/s) distribution through the scrubbing vanes of scrubber module A at 34 kg/s gas flow.

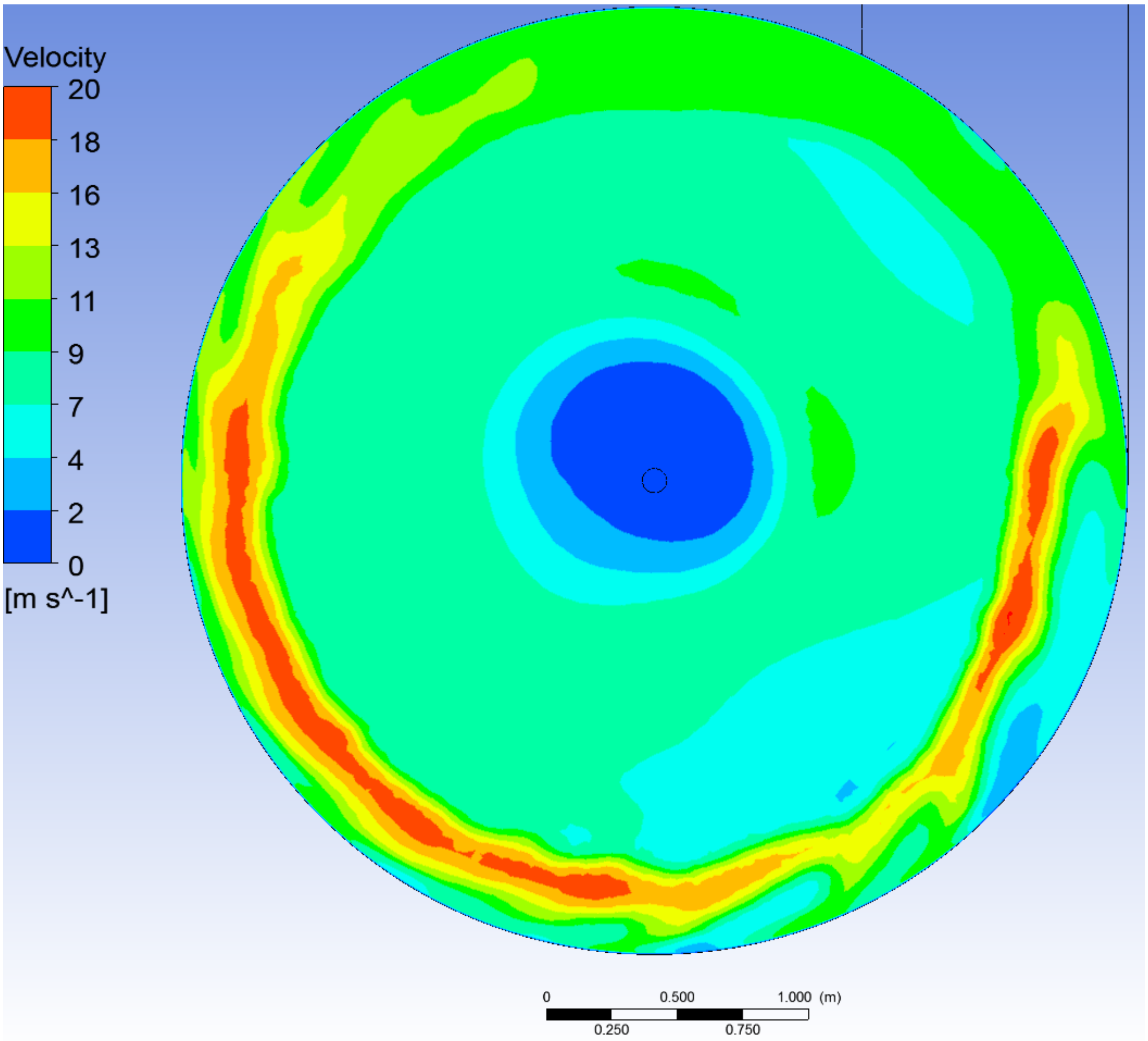


Figure 17 **Plan view of the predicted gas velocity (m/s) distribution above the water distribution cone of scrubber module A at 34 kg/s gas flow.**

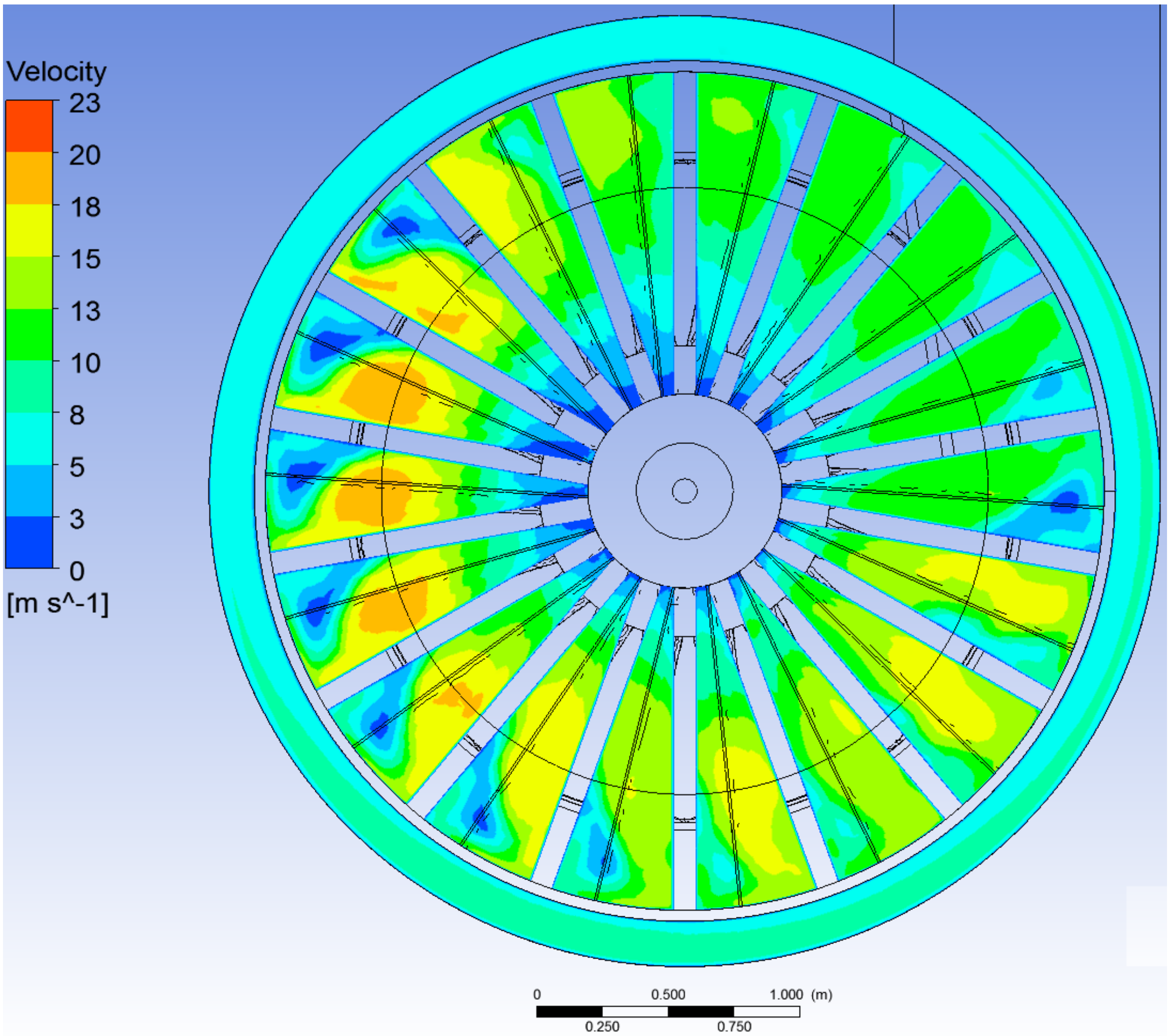


Figure 18 **Plan view of the predicted gas velocity (m/s) distribution through the demisting vanes of scrubber module A at 34 kg/s gas flow.**

Figure 19 shows plan views of the predicted gas velocity distribution through scrubber module B at four different gas flow rates. The modelled gas flow rate determines velocity magnitudes as expected but has little effect on the relative velocities; the locations of the high and low velocity zones are almost identical at the different gas flow rates. Furthermore, the predicted gas velocity distributions for scrubber module B are quite similar to those for scrubber module A (figure 16).

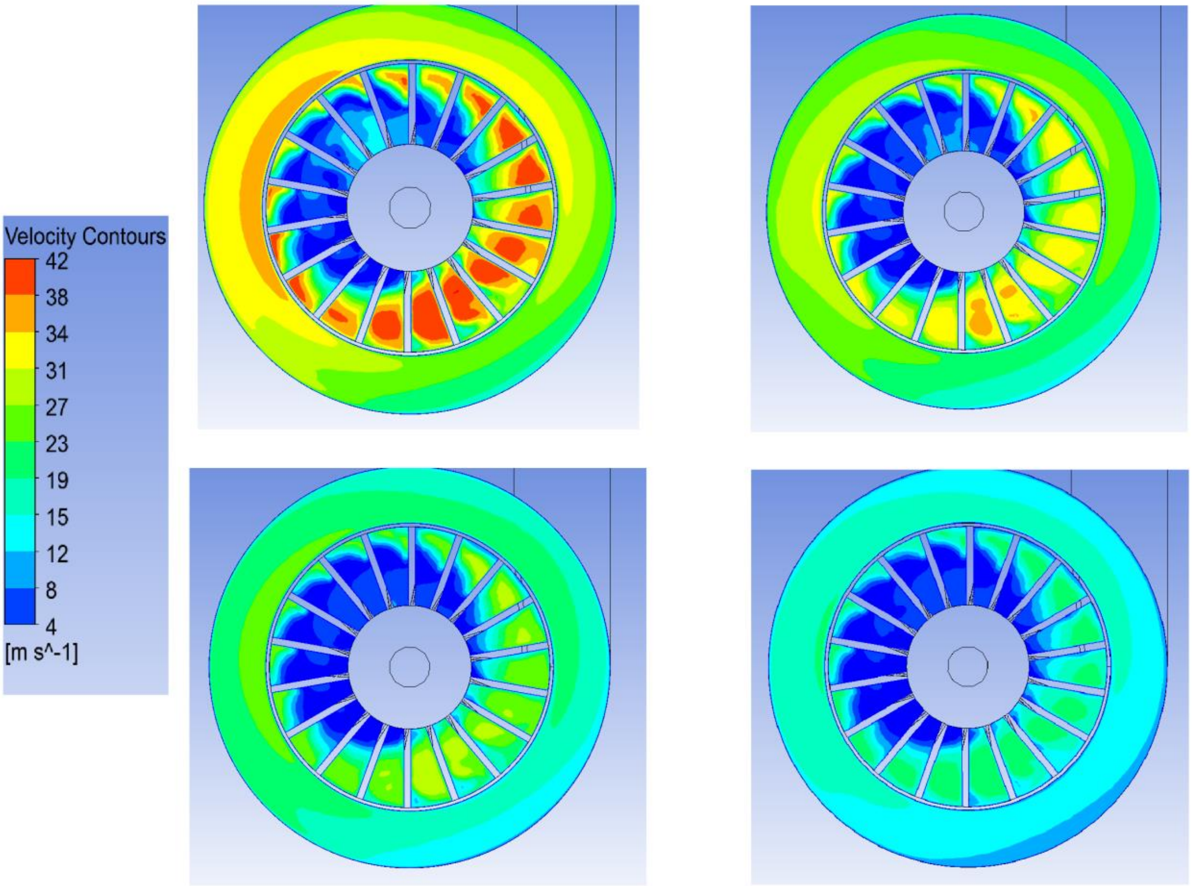


Figure 19 **Plan view of the predicted gas velocity (m/s) distribution through the scrubbing vanes of scrubber module B at (a) 34 kg/s, (b) 30 kg/s, (c) 24.25 kg/s and (d) 15 kg/s gas flow.**

Prediction of collection efficiency

The upgraded CFD code was used to model the air and water flows through the scrubber scale model. Once a converged solution for the air and water flows was obtained, dust particles with a size distribution measured at the inlet of a full scale dust collector, were introduced into the simulation. This process was repeated for different air flows and post processing was carried out to predict the collection efficiency for each of these air flows.

The predicted variations of overall collection efficiency with air flow through the scrubber scale model without and with droplet carryover being taken into account are compared in figure 20. The predicted droplet carryover, expressed as a percentage of the inlet water flow, is also included in the figure. The predicted collection efficiency without taking droplet carryover into account increases with the air flow over the range of air flows simulated but the rate of increase starts to taper off at air flows above 0.22 kg/s. When the reduction in collection efficiency due to droplet carryover is taken into account the collection efficiency of the SSM is predicted to decrease when the air flow exceeds 0.287 kg/s. This reduction in the predicted collection efficiency corresponds to the predicted droplet carryover exceeding 3% of the inlet water flow.

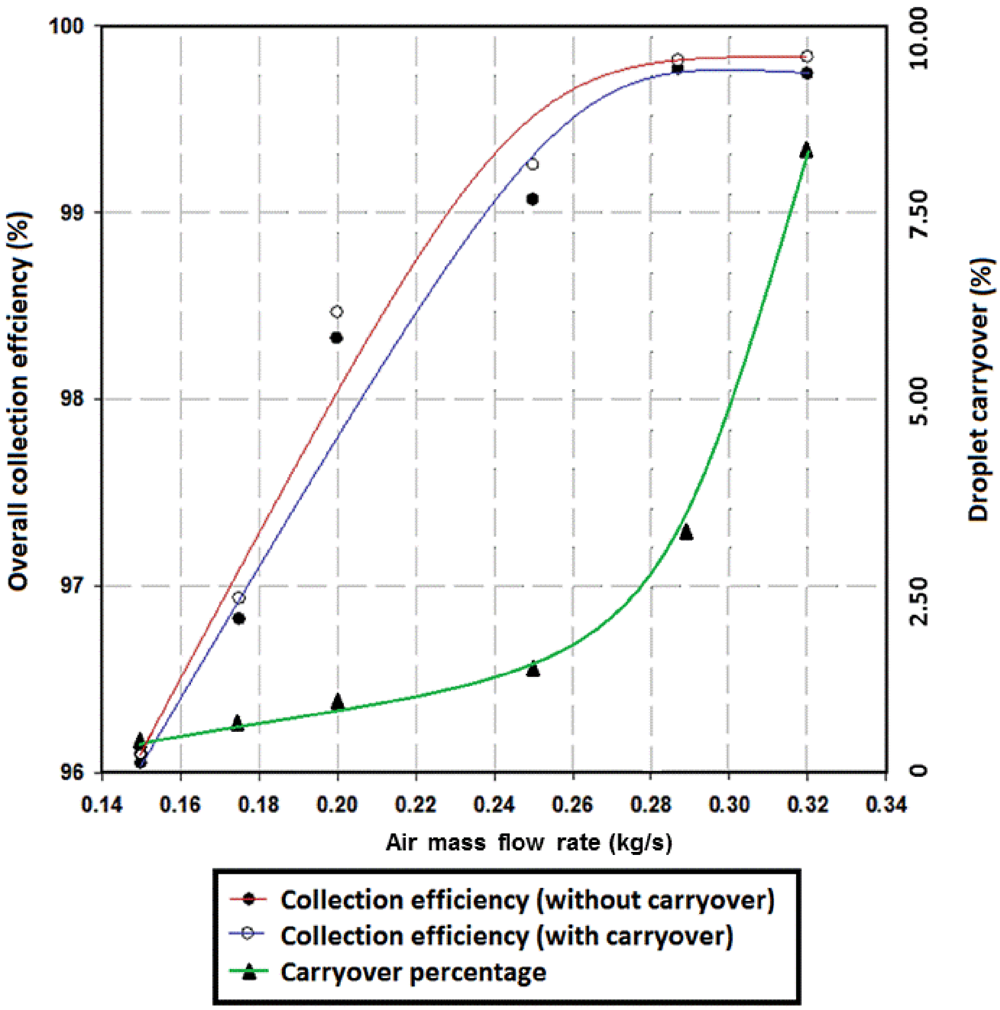


Figure 20 **Predicted variation of collection efficiency (without and with droplet carryover being taken into account) and droplet carryover with air mass flow rate in the scrubber scale model.**

Proposed design changes for improving scrubber performance

Based on observations during the scrubber scale model experiments and modelling of the scrubber scale model and full scale scrubber, several scrubber design changes were proposed which included:

- Raising the demisting vanes
- Installing a breakwater annulus between the scrubbing and demisting vanes and raising the demister vanes
- Installation of a vertical breakwater between the scrubbing and demisting vanes
- Addition of a drainage slot in in the annulus supporting the scrubbing vanes
- Addition of a vertical breakwater in the bottom cone of the scrubber

Modelling and/or modifications to the scrubber scale model were used to evaluate these design changes.

Raising the demister vanes

Droplet carryover can become excessive in a full scale scrubber when the agitated swirling water bath (figure 13) above the scrubbing vanes extends close to or beyond the demisting vanes.

Raising the demisting vanes makes it less likely for the water bath to extend past the demisting vanes. Eulerian modelling of the air and water flows through the scrubber scale model was carried out without and with the raised demister vanes. Figure 21 compares predicted volume fraction distribution of 550 μm diameter droplets in the scrubber scale model with the existing distance between the demisting and scrubbing vanes and with the distance between the two sets of vanes increased by raising the demisting vanes. The predicted volume fraction of droplets above the raised demisting vanes is significantly lower than the droplet concentration above the demisting vanes in the existing position. This suggests that droplet carryover should be reduced with the raised demisting vanes.

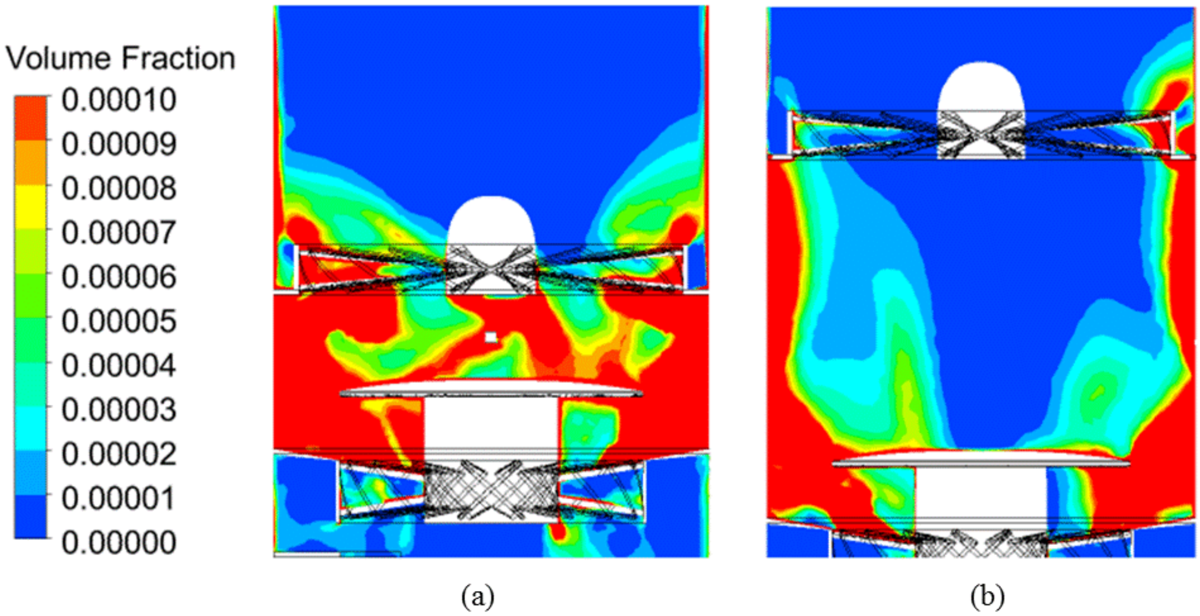


Figure 21 **Predicted contours of the volume fraction of 550 μm droplets in the scrubber scale model with (a) the existing spacing between the scrubbing and demisting vanes and (b) the proposed raised demisting vanes.**

Installing a breakwater annulus between the scrubbing and demisting vanes and raising the demister vanes

The breakwater annulus is intended to work by restricting the flow of water droplets up the inside wall of the scrubber towards the demister vanes. CFD simulations predicted and subsequent modifications to the scrubber scale model confirmed that provided the inside diameter of the breakwater was greater than the inside diameter of the annulus supporting the scrubbing vanes, installing the breakwater would not increase the pressure drop across the scrubber module. Figure 22 shows the predicted volume fraction distribution of 550 μm diameter droplets in the scrubber scale model with the proposed breakwater installed and the raised demister vane assembly. Very little carryover of 550 μm diameter droplets is predicted to occur with these changes.

This breakwater annulus and raising the demisting vanes modification was carried out on the scrubber scale model and the effects of these changes are shown in figure 23. With these modifications much less water enters the demisting vanes.

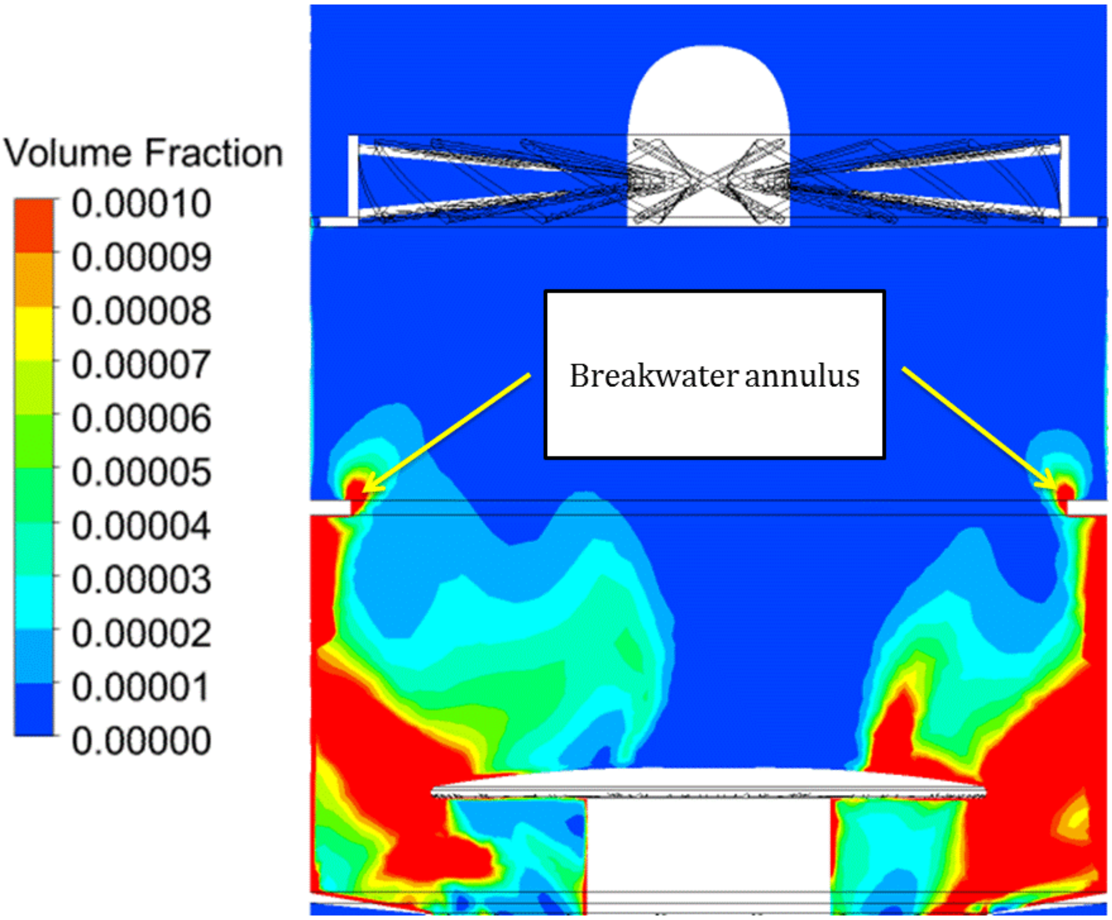


Figure 22 **Predicted contours of the volume fraction of 550 µm droplets in the scrubber scale model with the breakwater installed and the raised demister vane assembly.**

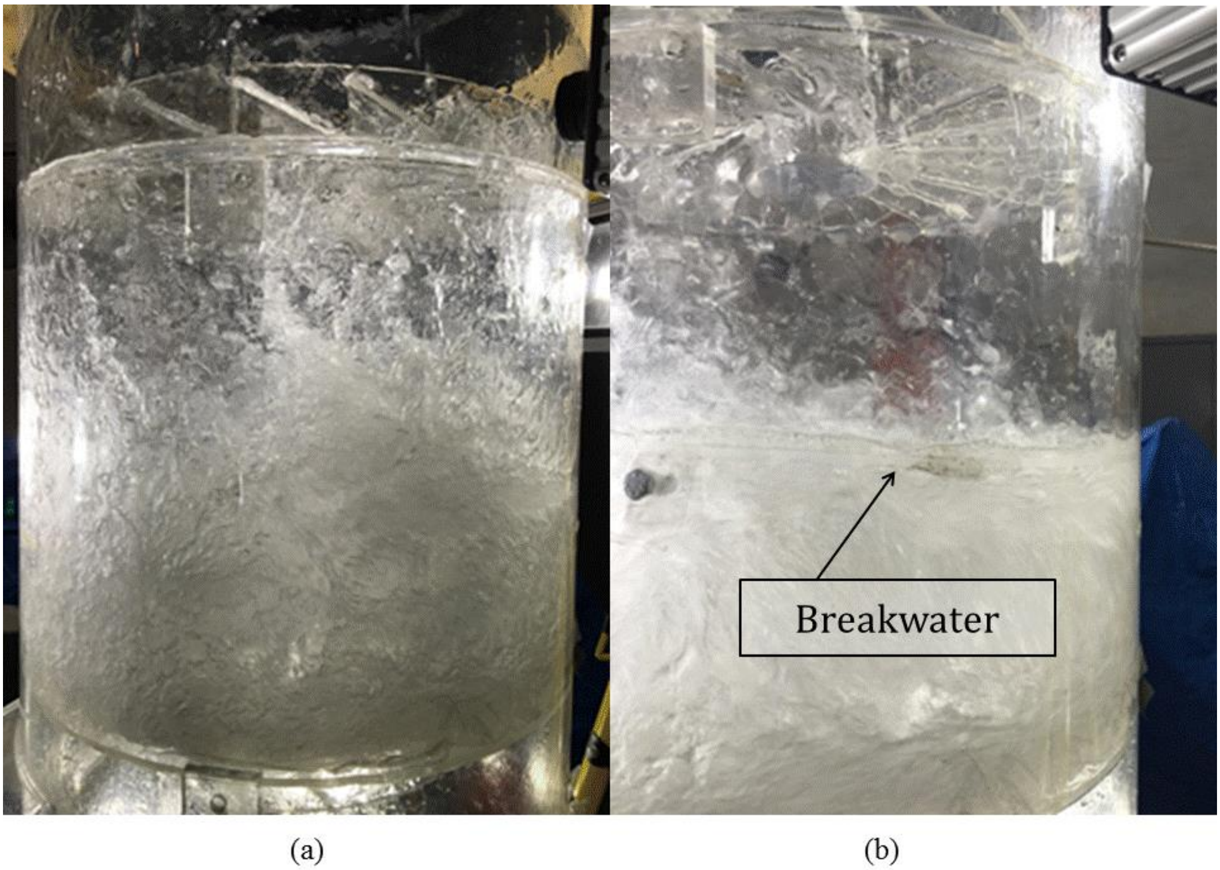


Figure 23 **Flow patterns in the scrubber scale model (a) before and (b) after the addition of the breakwater and the raising of the demister vanes.**

Installation of a vertical breakwater between the scrubbing and demisting vanes
Figure 24 shows the geometry of the vertical breakwater attached to the inside wall of the scrubber module. The purpose of this vertical breakwater is to disrupt the swirl of the water bath above the scrubbing vanes and force the water to run down along this breakwater back towards the central scrubbing vanes. The circumferential location of this breakwater should correspond to the area of the wall that faces the incoming flow. This will make more of the water running down through the scrubbing vanes come in direct contact with the incoming flue gas so that as much dust as possible can be removed from the gas prior to it entering the scrubbing vanes. This proposed modification was evaluated by modelling the gas flows through a full scale scrubber module with this modification. Figure 25 shows a plan view of the predicted gas velocity distribution in a plane between the scrubbing and demister vanes of a full scale scrubber module with the vertical breakwater.

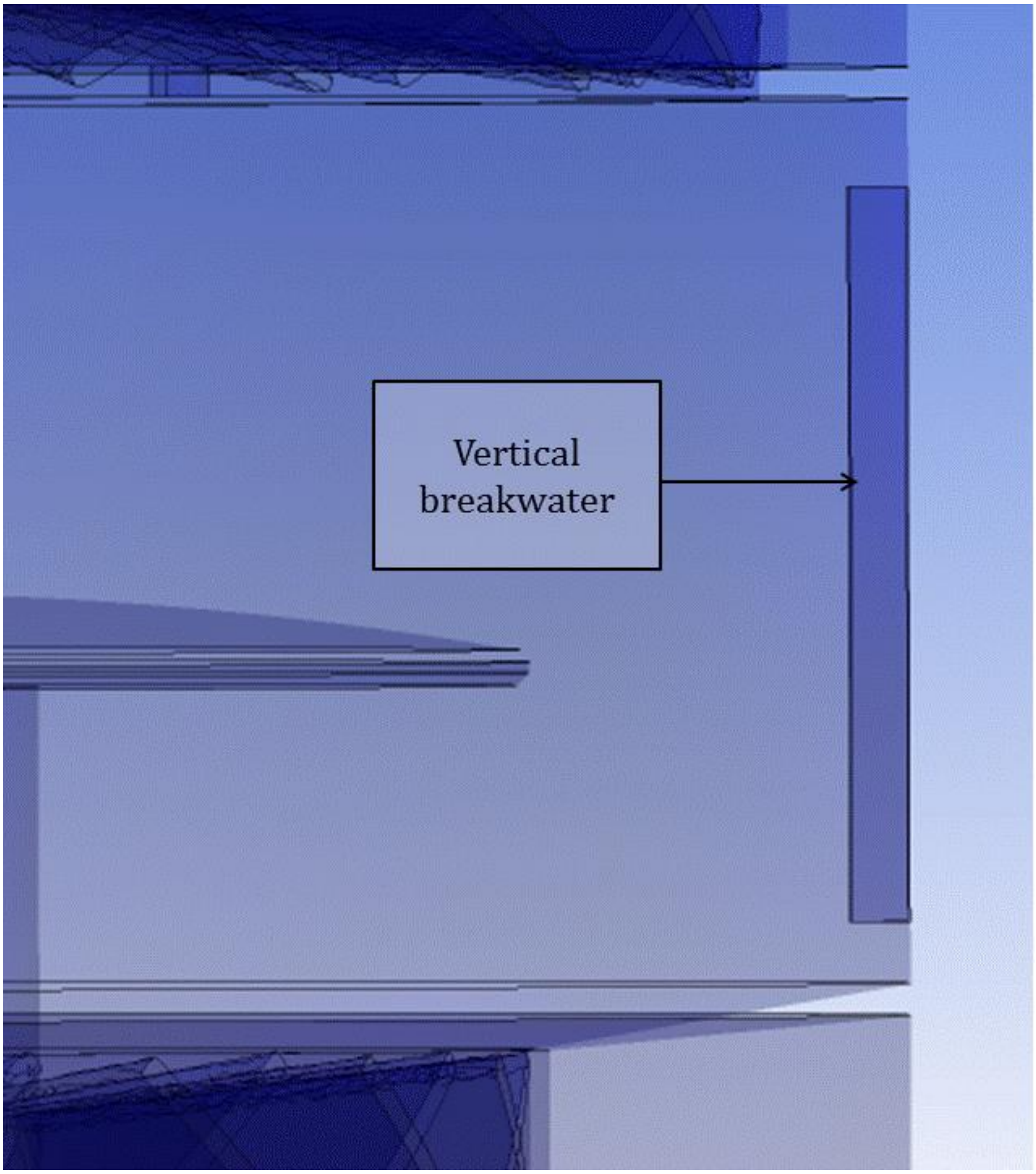


Figure 24 **The proposed vertical breakwater on the scrubber wall between the scrubbing and demisting vanes.**

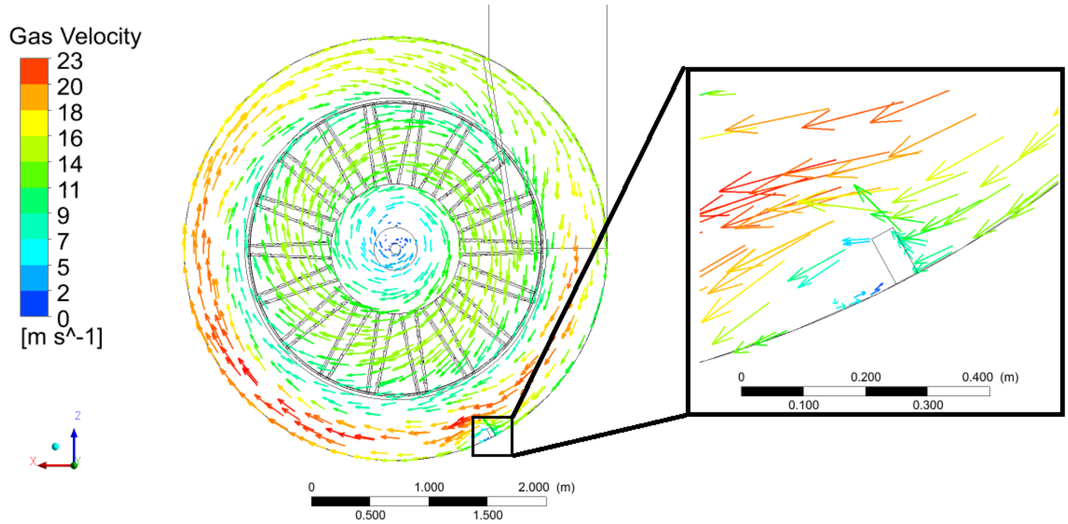


Figure 25 **Plan view of the predicted gas velocity distribution in a full scale scrubber module in a plane between the scrubbing and demisting vanes with the added vertical breakwater baffle.**

Addition of a drainage slot in in the annulus supporting the scrubbing vanes
To help reduce the level of the water bath between the scrubbing and demister vanes a drainage slot could be installed in the annulus supporting the scrubbing vanes. Figure 26 shows the predicted volume fraction distribution of 550 µm droplets in the scrubber scale model with such a slot installed. It is recommended that the circumferential location of any such slots should correspond to the area of the wall that faces the incoming gas flow. This will help ensure that there will be a substantial film of water on this part of the wall below the scrubbing vanes that can capture the incoming dust particles in a full scale scrubber module.

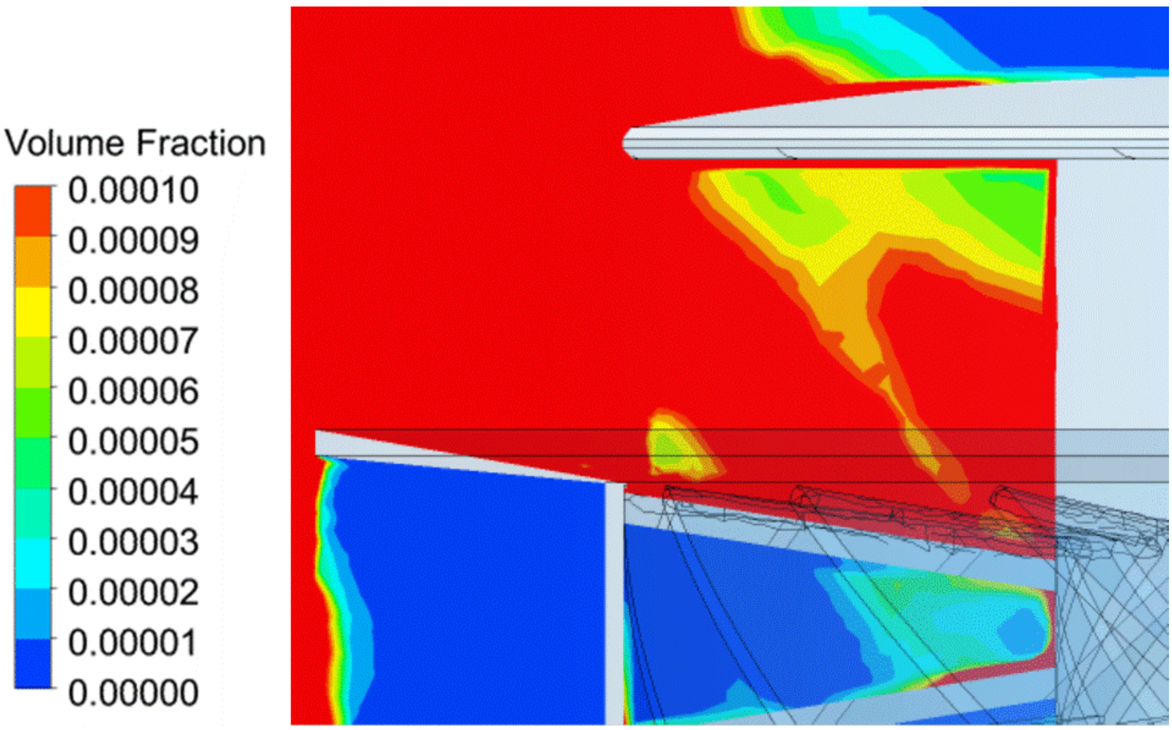


Figure 26 **Side elevation view of the predicted volume fraction of 550 µm droplets in the scrubber scale model with the slot installed in the annulus supporting the scrubbing vanes.**

Addition of a vertical breakwater in the bottom cone of the scrubber

During the scale model tests it was noted that water accumulates and swirls in the bottom cone which can cause water to splash into the inlet duct. In a full scale scrubber module this water would be dirty and could cause build-up in the scrubber inlet duct. The scrubber scale model was modified by installing a breakwater in the bottom cone that stops most of the swirl occurring and reduces the water level in the cone. Figure 27 shows water flowing in the bottom cone of the scrubber scale model with the breakwater plate installed.

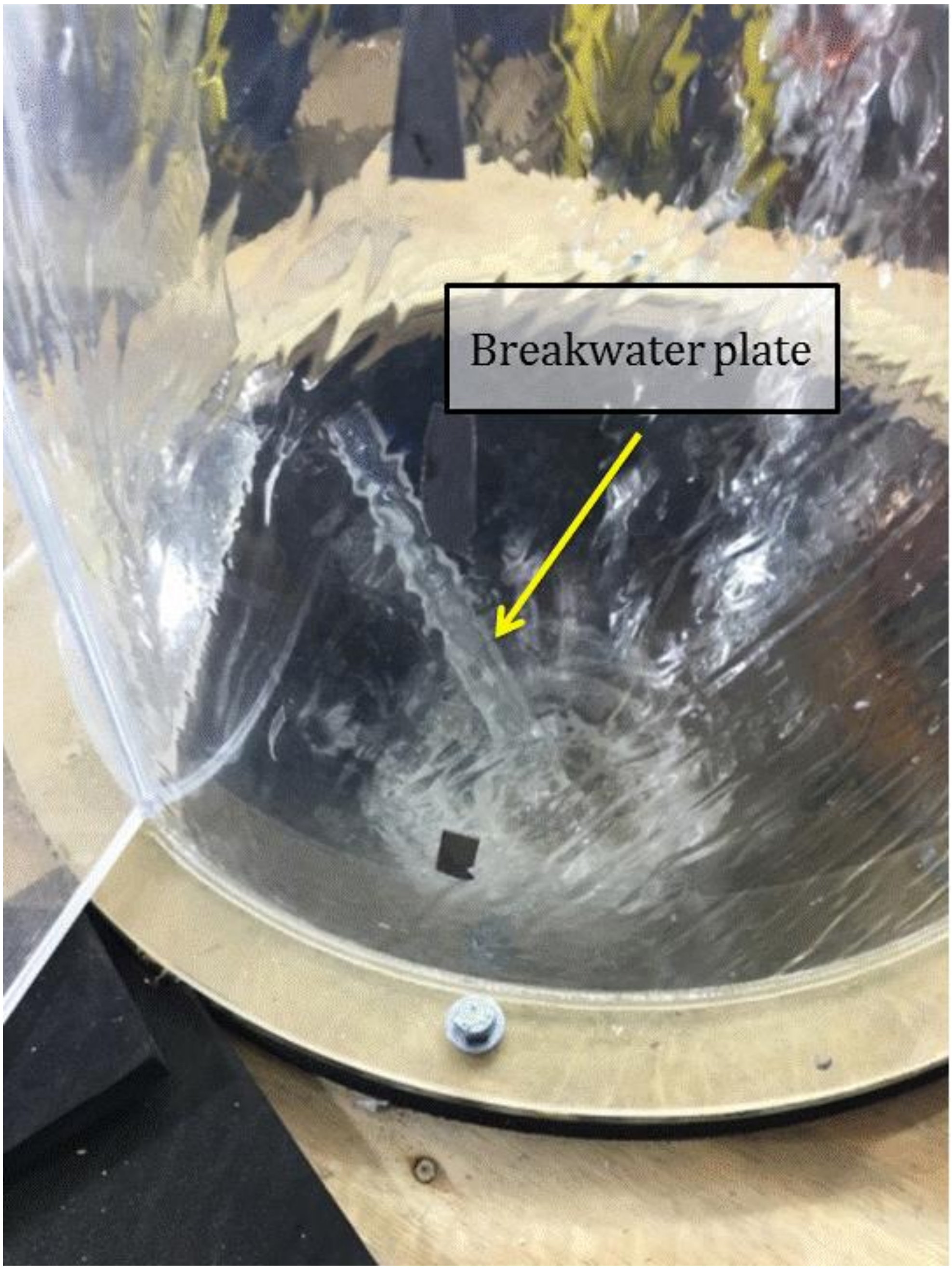


Figure 27 **Water flowing in the bottom cone of the scrubber scale model with the breakwater plate installed.**

Feedback from factory staff and a scrubber manufacturer

There was general agreement that the breakwater annulus and the raised demister vanes should reduce droplet carryover and that the vertical breakwater in the bottom cone should reduce swirl in that region. However it was pointed out that raising the demister vanes can be difficult to implement and could the option with only the breakwater annulus be considered.

The vertical breakwater proposal provoked a mixed response; one respondent thought it would reduce swirl which was undesirable while another thought a spiral vertical breakwater that would direct the water flowing down the wall into drainage slots would be more useful. Some respondents agreed with putting a drainage slot in the annulus supporting the scrubbing vanes and suggested that multiple drainage slots would be even better while other respondents did not think the drainage slots would be useful.

Updated modelling based on feedback

Figure 28 shows the predicted volume fraction distributions of 550 μm diameter droplets in the unmodified scrubber scale model and in the scrubber scale model with the breakwater installed and the height of the demister vanes unchanged. The predicted volume fraction of droplets above the demisting vanes and therefore droplet carryover is reduced with the breakwater installed. However the predicted reduction in the droplet volume fraction above the demister vanes is less than that predicted with the raised demister vanes and with the raised demister vanes and breakwater annulus combination.

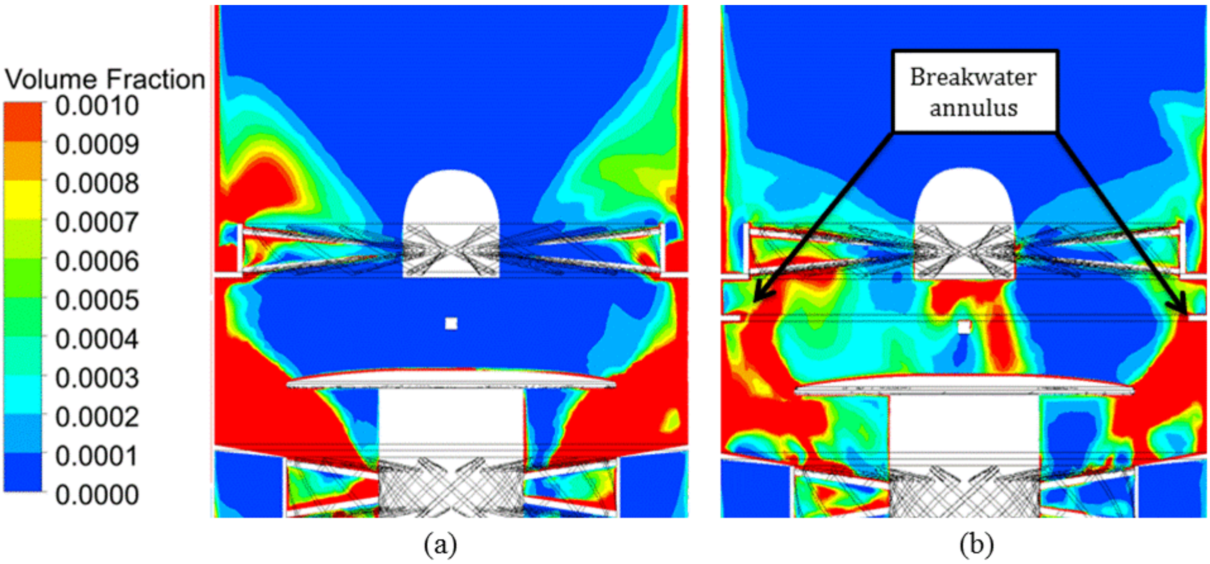


Figure 28 **Predicted contours of the volume fraction of 550 μm droplets in (a) the unmodified scrubber scale model and (b) the scrubber scale model with the breakwater annulus installed.**

A side elevation view of predicted air velocity vectors through the scrubber scale model with the breakwater annulus installed is shown in figure 29. Note the high velocity air flow from the scrubbing vanes to the right of the figure that is initially directed towards the wall before passing inside the breakwater annulus and then into the demister vanes. The upper limit of the swirling water bath is restricted to the height of the breakwater annulus. If the breakwater annulus is too low the high velocity air flow from the scrubbing vanes will be directed towards the scrubber scale model wall above the breakwater annulus and would not come in contact with the water bath. In a full scale scrubber this corresponds to dust laden gas flow not being collected by the scrubbing water and reduced collection efficiency.

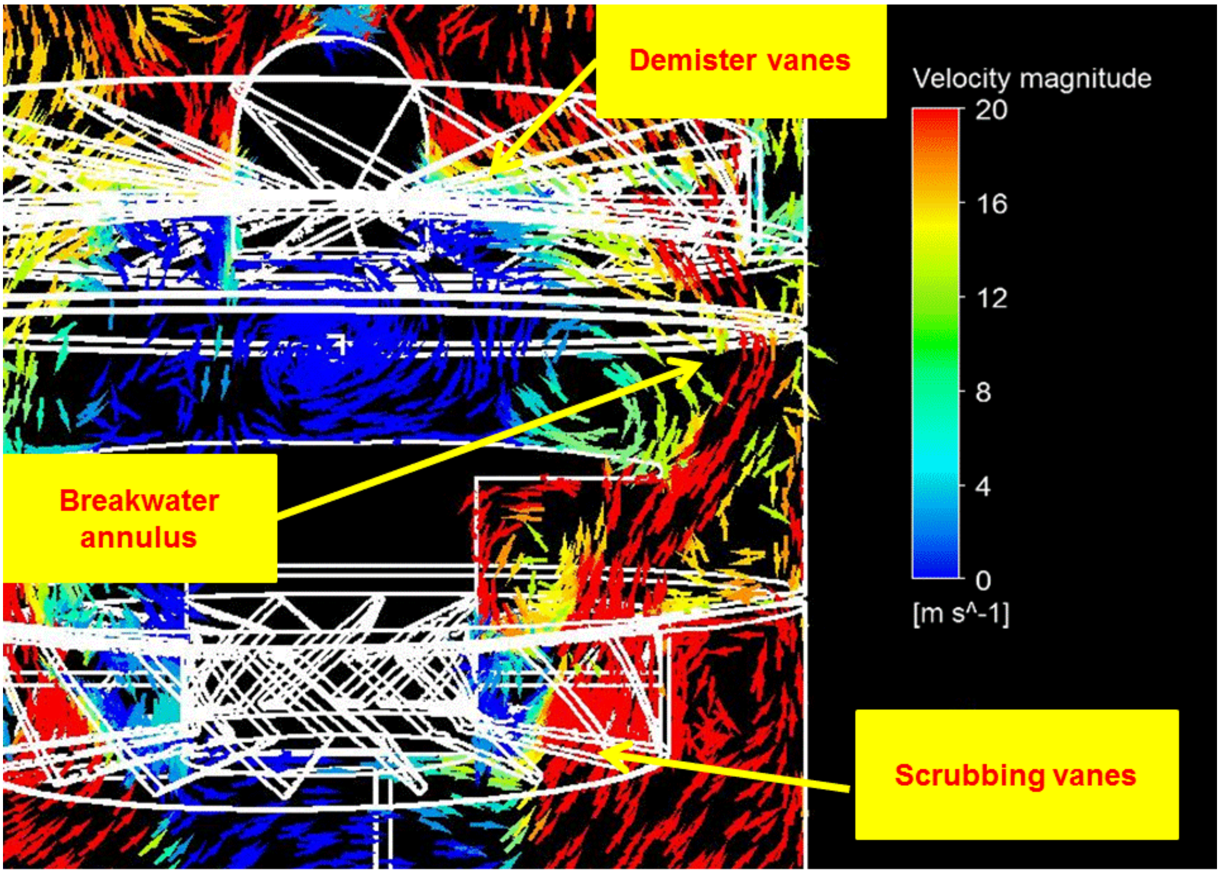


Figure 29 Side elevation view of the predicted air velocity vectors through the scrubber scale model with the breakwater annulus installed.

Note also from figure 29 that the direction of the air flow from the scrubbing vanes corresponds roughly to the angle of the scrubbing vanes. A projection drawn from a scrubbing vane to the wall at the angle of the scrubbing vane would give an estimate of the lowest vertical position of the breakwater annulus.

CFD modelling of the test rig scrubber scale model was carried out to predict the effect of the proposed vertical breakwater on the water and air flows. Figure 30 shows a side elevation view of the iso-surface in the scrubber scale model where the water volume fraction was predicted to be equal to 0.001 with the proposed vertical breakwater installed. More water is predicted to flow down through the scrubbing vanes on the side of the scrubber scale model with the vertical breakwater installed. The vertical breakwater is on the opposite site of the scale model to the inlet duct so that the incoming air flow is directed towards the water draining through the scrubbing vanes. This corresponds to the incoming dust laden gas flow in a full scale scrubber being directed towards where the water drains through the scrubber vanes for improved mixing between the dust particles and the water and therefore improved collection efficiency.

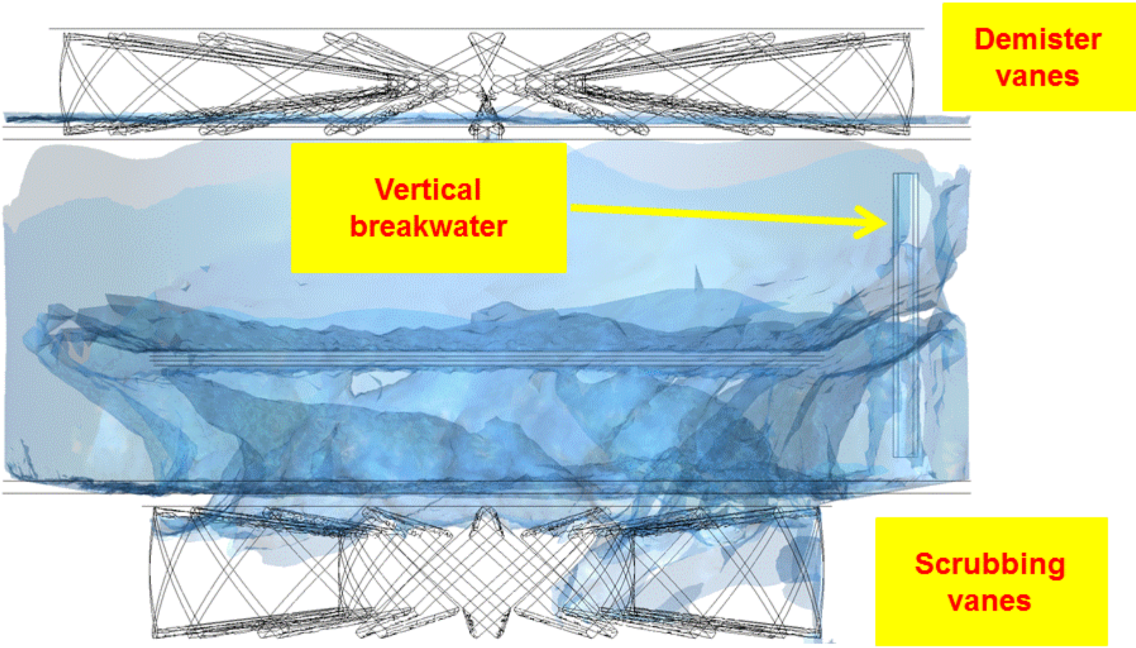


Figure 30 **Side elevation view of the iso-surface in the scrubber scale model where the water volume fraction was predicted to be equal to 0.001 with the proposed vertical breakwater installed.**

One of the concerns raised about the vertical breakwater proposal was the possibility that it would reduce the swirl in the flow between the scrubbing and demister vanes. Figure 31 shows a part plan view of the predicted air and droplet velocity vectors through the scrubber scale model with the vertical breakwater installed. There is only a small disruption predicted to the main swirling air flow (black velocity vectors) that occurs around the vertical breakwater. It is therefore unlikely that a vertical breakwater of the modelled relative size will have much effect on the overall swirling flow pattern. Figure 31 also shows that the droplet velocity vectors have a large radial velocity component while the air velocity vectors have virtually no radial velocity component.

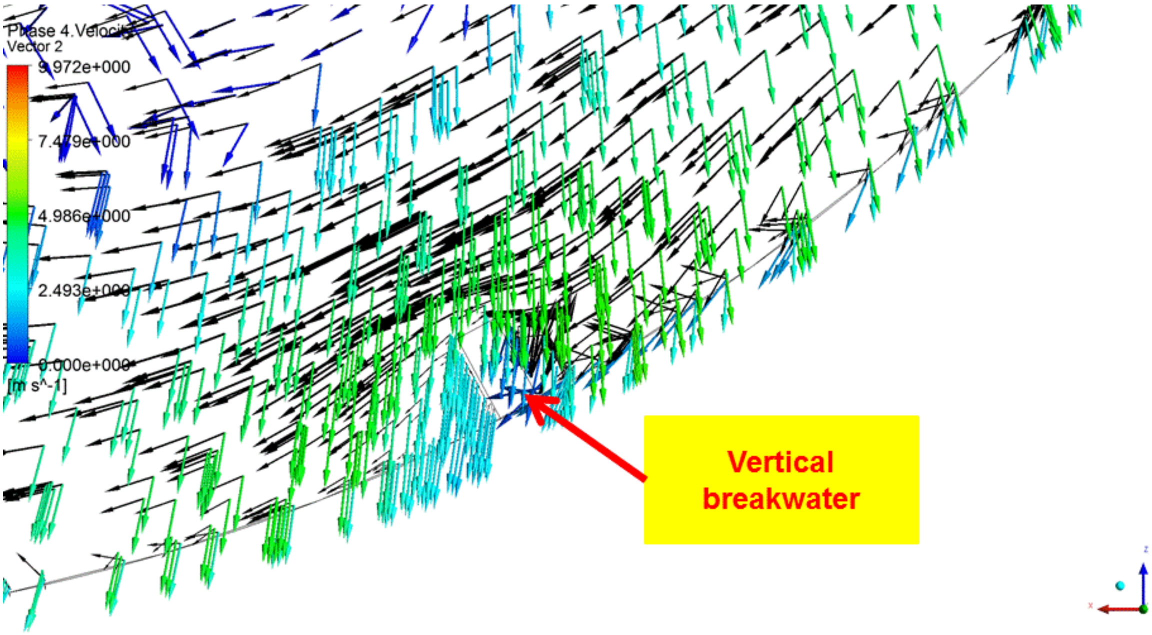


Figure 31 Part plan view of the predicted air (in black) and droplet (other colours) velocity vectors between the scrubbing and demisting vanes of the scrubber scale model with the proposed vertical breakwater installed.

Recommended scrubber design changes

From the modelling and experimental work carried out in this project the combination of raising the demister vanes and installing a breakwater annulus should significantly reduce droplet carryover. The modelling indicates that raising the demister vanes would be the next most effective option for reducing droplet carryover followed by the annular breakwater between the scrubbing and demister vanes. It is noted that if the breakwater is too close to the scrubbing vanes collection efficiency could reduce. There was general agreement between the factory staff and the scrubber manufacturer that these modifications should reduce droplet carryover.

When a vertical breakwater was installed in the bottom cone of the test rig scale model, the water drained more quickly as the breakwater reduced the swirl. This suggests that a vertical breakwater installed in the bottom cone of a full scale scrubber module should reduce the swirl and help the dirty water in the cone drain more quickly. There was general agreement that this modification would be useful. If the bottom cone of a full scale scrubber can drain more quickly this should reduce the amount of dirty water splashing up into the gas inlet duct and therefore reduce inlet duct blockages.

Even though it received mixed feedback, a vertical breakwater installed between the scrubbing vanes and the demister vanes may also be worth considering because it is predicted to help drain some of the water from between the scrubbing and demisting vanes which will help when the level of the swirling water bath gets too high. It should also direct more water into the path of the incoming dust laden gas flow which should improve collection efficiency.

Discussion

This project has achieved all its objectives. An improved understanding of the flow processes occurring in centrifugal wet scrubbers has been obtained. Based on the knowledge gained and the CFD modelling carried out in this project, design changes for improved scrubber performance have been recommended.

The air flow only tests on the scrubber scale model and the associated modelling identified the strong non symmetrical flow patterns and the low velocity zones that correspond to build up in full scale scrubber modules. The predicted air flow distributions through the scrubber scale model were similar to the predicted gas flow distribution in the modelled full scale scrubber modules.

The tests on the scrubber scale model with both air and water flow showed how a swirling water bath forms between the scrubbing and demisting vanes. At low air flows the water bath did not form because the air velocities in the scrubber scale model were not sufficient to prevent water draining back down through the scrubbing vanes. At very high air flows the water bath extended into the demister vanes which resulted in excessive droplet carryover. This is consistent with the increased droplet carryover that occurs in a full scale scrubber module when the gas flows are very high. It is likely that the water bath captures the dust particles that pass through the scrubbing vanes. Without the water bath, this capture of dust particles that pass through the demister vanes is unlikely to occur which is consistent with the lower collection efficiency of wet centrifugal scrubbers at low loads. The CFD modelling of the air and water flows through the scrubber scale model with different air flows produced results that were similar to the observations. The high speed photography identified the different stages of water curtain break-up. Agglomeration of water droplets was observed to occur primarily at the scrubber wall which is mostly covered by a film formed by these droplets below the scrubbing vanes.

The CFD code was updated to predict the capture of dust particles by water droplets and the wall film. The model updates included a kinetic energy test for dust particle capture that takes into account the velocity of the dust particle relative to the water and the water surface tension. With the upgraded CFD code the collection efficiency of the scrubber scale model could be predicted for a range of air flows. The collection calculations were adjusted to take into account the reduction in collection efficiency due to droplet carryover. With the updated collection efficiency calculation the air flow rate at which collection efficiency is maximised could be determined. Many other collection efficiency models used in the literature do not take droplet carryover into account and will therefore predict that collection efficiency always increases with gas flow, which is not the case in practice.

Observations from the scrubber scale model experiments and associated CFD modelling formed the basis of proposed design changes. Further modelling and modifications to the scrubber scale model were made to evaluate the proposed design changes. The feedback from factory staff and the scrubber manufacturer on the proposed design changes and further modelling was carried out based on this feedback. This lead to updated scrubber design change recommendations.

Acknowledgements

The input and assistance provided by Bingera Mill and Isis Mill staff during the scrubber inspections is greatly appreciated. The input and assistance provided by Neil McKenzie and Barry Hume with the experimental measurements and flow visualisation is gratefully acknowledged. The authors appreciate the feedback from the factory and Fowlerex staff on the proposed scrubber design modifications and the computing support provided by QUT high performance computing. QUT is acknowledged for funding the attendance by Hassan Ali at the Australian Heat and Mass Transfer Conference at QUT and the 11th European Fluid Mechanics Conference in Seville.

References

Adeniji-Fashola A, Chen CP (1990) Modeling of confined turbulent fluid-particle flows using Eulerian and Lagrangian schemes. *International Journal of Heat and Mass Transfer* **33**, 691-701.

Ahmadvand F, Talaie MR (2010) CFD modeling of droplet dispersion in a Venturi scrubber. *Chemical Engineering Journal* **160**, 423-431.

Ali M, Yan C, Sun Z, Gu H, Mehboob K (2015) Dust particle removal efficiency of a venturi scrubber. *Annals of Nuclear Energy* **54**, 178-183.

Bird RB, Stewart WE, Lightfoot EN (1960) 'Transport phenomena'. Wiley New York.

Boyd RK (1986) Computer modelling of a coal fired furnace. PhD thesis, University of Sydney, Sydney.

Boysan F, Ayers WH, Swithenbank J (1982) A fundamental mathematical modelling approach to cyclone design. *Transactions of the Institution of Chemical Engineers* **60**, 222-230.

Calvert S (1970) Venturi and other atomizing scrubbers efficiency and pressure drop. *AIChE Journal* **16**, 392-396.

Calvert S, Goldshmid J, Leith D (1974) Scrubber performance for particle collection. *AIChE Symp. Ser* **72**, 357-364.

Chuah TG, Gimbun J, Choong TSY (2006) A CFD study of the effect of cone dimensions on sampling aerocyclones performance and hydrodynamics. *Powder Technology* **162**, 126-132.

Cortes C, Gil A (2007) Modeling the gas and particle flow inside cyclone separators. *Progress in Energy and Combustion Science* **33**, 409-452.

Costa M, Riberio A, Tognetti E, Aguiar M, Gonclaves J, Coury J (2005) Performance of a venturi scrubber in the removal of fine powder from a confined gas stream. *Materials Research* **18**.

Crowe CT (1982) Review - Numerical models for dilute gas-particle flows. *Journal of Fluids Engineering* **104**, 297-303.

Crowe CT, Pratt DT (1974) Analysis of the flow field in cyclone separators. *Computers & Fluids* **2**, 249-260.

Crowe CT, Sharma MP, Stock DE (1977) The particle-source-in-cell (PSI-Cell) model for gas-droplet flows. *Journal of Fluids Engineering* **99**, 325-332.

Dixon TF, Mann AP, Plaza F, Gilfillan WN (2005) Development of advanced technology for biomass combustion - CFD as an essential tool. *Fuel* **84**, 1303-1311.

Downing CM, Dixon TF, Pennisi SN (2000) Computational fluid dynamics modelling of HMA Ducon wet scrubber - poster paper. *Proceedings of the Australian Society of Sugar Cane Technologists* **22**, 505-531.

Drew DA (1983) Mathematical modeling of two-phase flow. **15**, 261-291.

Dukowicz JK (1980) A particle-fluid numerical model for liquid sprays. *Journal of Computational Physics* **35**, 229-253.

Durst F, Miloievic D, Schonung B (1984) Eulerian and Lagrangian predictions of particulate two-phase flows: a numerical study. *Applied Mathematical Modelling* **8**, 101-115.

Gosman AD, Ioannides E (1981) Aspects of computer simulation of liquid-fueled combustors. *AIAA* **81-0234**.

Griffiths WD, Boysan F (1996) Computational fluid dynamics (CFD) and empirical modelling of the performance of a number of cyclone samplers. *Journal of Aerosol Science* **27**, 281-304.

Guerra VG, Béttega R, Gonçalves JAS, R. CJ (2012) Pressure drop and liquid distribution in a venturi scrubber: experimental data and CFD simulation. *Industrial and Chemical Engineering Research* **51**, 8049-8060.

Harlow FH, Amsden AA (1975) Numerical calculation of multiphase fluid flow. **17**, 19-52.

Hinze J (1972) 'Turbulence'. McGraw, New York.

Hounslow MJ, Ryall RL, Marshall VR (1988) A discretized population balance for nucleation, growth, and aggregation. *AIChE Journal* **34**, 1821-1832.

Jones WP (1949) Development of the Venturi scrubber. *Industrial and Engineering Chemistry* **41**, 2424-2427.

Karagoz I, Kaya F (2007) CFD investigation of the flow and heat transfer characteristics in a tangential inlet cyclone. *International Communications in Heat and Mass Transfer* **34**, 1119-1126.

Lee B-K, Mohan BR, Byeon S-H, Lim K-S, Hong E-P (2013) Evaluating the performance of a turbulent wet scrubber for scrubbing particulate matter. *Journal of the Air & Waste Management Association* **63**, 499-506.

Leonard G, Mitchner M, Self SA (1980) Particle transport in electrostatic precipitators. *Atmospheric Environment* **14**, 1289-1299.

Lockwood FC, Salooja AP, Syed SA (1980) A prediction method for coal-fired furnaces. *Combustion and Flame* **38**, 1-15.

Lora ES, Jativa IH (1999) Air pollution control in the sugar cane industry. *Proceedings of the XXIII Congress, International Society of Sugar Cane Technologists*321-330.

Luo M, Stanmore BR (1994) Modelling combustion in a bagasse-fired furnace. 1: Formulation and testing of the model. *Journal of the Institute of Energy* **67**, 128-135.

Mann AP (1996) Applications of computational furnace modelling. PhD thesis, University of Sydney, Sydney.

Melville WK, Bray KNC (1979) A model of the two-phase turbulent jet. **22**, 647-656.

Migdal D, Agosta VD (1967) A source flow model for continuum gas-particle flow. *Journal of Applied Mechanics* **34**, 860-865.

Mohebbi A, Taheri M, Fathikaljahi J, Talaie MR (2003) Simulation of an orifice scrubber performance based on Eulerian/Lagrangian method. *Journal of Hazardous Materials* **100**, 13-25.

Moor BSC (2007) Flue gas scrubbing equipment for bagasse fired boilers. *Proceedings of the XXVI Congress, International Society of Sugar Cane Technologists*1273-1283.

Narasimha M, Brennan MS, Holtham PN, Napier-Munn TJ (2007) A comprehensive CFD model of dense medium cyclone performance. *Minerals Engineering* **20**, 414-426.

O'Rourke PJ (1981) Collective drop effects on vaporizing liquid sprays. PhD thesis, Princeton University.

Orme M (1997) Experiments on droplet collisions, bounce and coalescence. *Progress in Energy and Combustion Science* **23**, 65-79.

Pak SI, Chang KS (2006) Performance estimation of a Venturi scrubber using a computational model for capturing dust particles with liquid spray. *Journal of Hazardous Materials* **138**, 560-573.

Palaszewski SJ, Jiji LM, Weinbaum S (1981) A three-dimensional air-vapor-droplet local interaction model for spray units. *Journal of Heat Transfer* **103**, 514-521.

Pan KL, Law CK (2007) Dynamics of droplet-film collision. *Journal of Fluid Mechanics* **587**, 1-22.

Patankar SV, Spalding DB (1972) A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *International Journal of Heat and Mass Transfer* **15**, 1787-1806.

Pemberton C (1960) Scavenging action of rain on nonwettable particulate matter suspended in the atmosphere. *Intern. J. Air Polution* **3**.

Pennington I (1999) Controlling particulate emission levels of bagasse fired boilers to meet the new DENR regulations. *Philsutech*.

Pilch M, Erdman CA (1987) Use of breakup time data and velocity history data to predict the maximum size of stable fragments for acceleration-induced breakup of a liquid drop. *International Journal of Multiphase Flow* **13**, 741-757.

Roache PJ (1976) 'Computational fluid dynamics'. Hermosa, Albuquerque.

Schifftner KC, Hesketh HE (1983) 'Wet scrubbers'. Ann Arbor Science Publishers Inc., Ann Arbor, Michigan.

Weber R, Visser BM, Boysan F (1990) Assessment of turbulence modeling for engineering prediction of swirling vortices in the near burner zone. *International Journal of Heat and Fluid Flow* **11**, 225-235.

Whitaker S (1973) The transport equations for multi-phase systems. **28**, 139-147.

Woodfield PL (2001) Combustion instability in bagasse-fired furnaces. PhD thesis, University of Sydney, Sydney.

Wu P-K, Kirkendall KA, Fuller RP, Nejad AS (1997) Breakup processes of liquid jets in subsonic crossflows. *Journal of Propulsion and Power* **13**, 64-73.

Outputs and Outcomes

Outputs

- Test rig and scrubber scale model
- Improved modelling capability
- Updated CFD model

- CFD modelling predictions
- Flow visualisation images of the water flows in the scrubber scale model showing the water curtain break-up mechanisms
- Proposed scrubber design changes and the associated modelling and scrubber scale model modifications that have been communicated to factory staff and the scrubber manufacturer Fowlerex
- Recommended scrubber design changes

Outcomes

- Reduced factory down time due to fewer stoppages for cleaning
- Increased factory throughput by reducing steam output limitations imposed by scrubber blockages
- Reduced emissions from sugar factory boilers
- Reduced complaints from the local community

Benefits

- Increased profitability of the sugar manufacturing process
- Reduced environment impact of the sugar industry

Intellectual Property (IP) and Confidentiality

No protectable IP was generated in this project.

Industry Communication and Adoption of Outputs

What key messages have come from the project to date, when and how they have been communicated and to whom? Has there been any communication with the SRA communication team or adoption team?

Project progress has been reported at the 2014, 2015, 2016 and 2017 regional research seminars. Initial measurements and comparisons with model predictions were presented at Hassan Ali's PhD confirmation seminar on the 9 February 2015 to QUT staff and students. Webinars on the project organised through Andrea Evers were completed on the 10 September 2014 and 22 February 2017. The paper 'Inside a wet scrubber' was presented at the 2016 ASSCT conference and was well received. Modelling and experimental results from the project were presented at the Australian Heat and Mass Transfer Conference at QUT (July 2016) and at the 11th European Fluid Mechanics Conference in Seville (September 2016).

What new information, if any, is available on the adoption of project outputs?

There was widespread interest in implementing some of the modelled design changes. Two of the recommended scrubber design changes: the vertical breakwater in the bottom cone and a drainage slot in the annulus supporting the scrubbing vanes, were implemented in two of the four scrubber modules on the Macknade No. 7 boiler. Initial reports from factory staff suggest that there is reduced carryover from the two modules where the design changes were implemented but this needs further follow up to confirm.

List any newsletters, fact sheets or any other media coverage.

There was an article on the project in issue 2 of Milling Matters in 2014.

Identify any further opportunities to disseminate and promote project outputs at seminars, field days etc.

It is intended that a wrap up of the project will be presented at the 2018 regional research seminars.

Environmental Impact

There was no environmental impact associated with carrying out the project. It is likely that the scrubber design change recommendations, when implemented, will reduce the environmental footprint of the sugar industry.

Recommendations and Future Industry Needs

Further follow up on the effects of the implemented design changes on operating scrubbers is recommended.

Publications

The 2016 ASSCT publication from this project 'Inside a wet scrubber' is included in the appendix. Two manuscripts based on the work in this project: 'Flow visualization and modelling of scrubbing liquid flow patterns inside a centrifugal wet scrubber' and 'Numerical prediction of dust capture efficiency of a centrifugal wet scrubber' have been submitted to international journals and are currently under review.

APPENDIX

2016 ASSCT paper 'Inside a wet scrubber'

M 6. INSIDE A WET SCRUBBER

By

H ALI, AP MANN, F PLAZA

Queensland University of Technology
a.mann@qut.edu.au

KEYWORDS: Wet Scrubber, Boiler, Collection Efficiency, Carryover

Abstract

Wet scrubbers are used in a range of industries for exhaust gas cleaning and are the Australian sugar industry's most effective means of removing dust from boiler flue gas, being responsible for significantly reducing its environmental footprint.

In virtually all cases, the collection efficiency of wet scrubbers is high enough for boilers with them installed to comfortably comply with the dust emission limits set by environmental authorities. However, wet scrubbers often cause boiler operational problems such as excessive vibration, deposit build-up and wear of boiler induced draft fans due to water droplet carryover and reduced boiler steam output due to blockages in and/or around the scrubber. In some cases, blockages become so severe that a boiler has to be taken off line for scrubber cleaning. Sugar milling operations are adversely affected by the reduced throughput and/or factory stoppages caused by these problems.

This paper describes modelling and experimental investigations carried out as part of a project that aims to improve the design and operation of the tangential entry fixed vane wet scrubbers that are widely used in the Australian sugar industry.

Introduction

During the past two decades, low cost, high speed computing resources have become more widely available. This has helped establish computational fluid dynamics (CFD) as an important tool for studying and improving a wide range of industrial fluid flow processes. CFD can be used to predict the flow of a single fluid such as air through a building or around an aircraft wing and the results used for design optimisation. When sub-models for combustion (single and multi-phase) and heat transfer (convection, conduction and radiation) are incorporated into CFD codes, the flow and temperature distributions can be predicted with sufficient accuracy to be used as a basis for design modifications in boilers (Mann and Rasmussen, 2011; Plaza *et al.*, 1999). However the flows in a wet scrubber are especially complex due to the interaction between dust particles, water droplets, bulk water and the turbulent flue gas flow. Further refinement of CFD codes is required to gain a better understanding of the important flow processes inside a wet scrubber. This improved understanding provides the basis for proposing design changes that can be evaluated with the upgraded CFD code.

For the results of CFD simulations to be used with confidence, experimental validation is required. This paper describes the experimental work that has been conducted to validate single phase CFD predictions and to visualise the gas, particle and water flow patterns. Preliminary multi-phase predictions are also presented.

Background

Dry cyclones and wet venturi scrubbers have been extensively studied but there are very few published investigations of the flow patterns in the tangential entry fixed vane wet scrubbers commonly used in the Australian sugar industry. Experimental work on dry cyclones that includes the measurement of collection efficiency and comparisons with empirical correlations has been carried out by Dirgo and Leith (1985). Akiyama and Marul (1986) carried out tests to determine the effect of guide vane geometry on dry cyclone collection efficiency. Numerical and experimental studies carried out during the development of a high efficiency, high temperature (>400 °C) dry cyclone are reported by Shin *et al.* (2005) while others (Kim and Lee, 1990) have carried out experiments using a glass scale model to determine the effect of flow rates and cyclone body dimensions on the collection efficiency.

Investigation into wet venturi scrubbers include the collection efficiency tests of Costa *et al.* (2005) who operated a test scale venturi scrubber with mineral salt as the dispersed phase. Collection efficiencies were measured for different scrubbing liquid flow rates and air velocities. Experimental results and CFD predictions were compared by Guerra *et al.* (2012); this work showed how scrubber pressure drop is affected by the number of liquid injection orifices. Ali *et al.* (2015) have measured the collection efficiency of a venturi scrubber using a filtration technique and compared the results with a previously developed expression for collection efficiency via inertial impaction (Calvert, 1970), which is usually the most important mechanism for dust collection in wet scrubbers (Pak and Chang, 2006).

Experimental test program

Test rig

An acrylic scale model of a wet scrubber module and inlet duct (approximately one tenth the full size) was constructed. The scale model included the scrubbing vanes, water distribution cone (with trough) and demister vanes. The outlet of this scale model was connected to an extraction fan (1.5 kW) with flexible and rigid ducting as shown in Figure 1. The air flow through the scale model was varied by controlling the fan speed with a three-phase variable speed drive. Holes were drilled through the wall and inlet duct of the scale model at various locations to facilitate measurement of air velocities and static pressures. A water reservoir was installed at ground level from where the water was pumped into the scrubber scale model through an inlet pipe that terminated just above the distribution cone. When water flows into the scale model it fills, then overflows the trough on the water distribution cone. A high speed camera (X-stream XS-4) together with the software Motion Studio was used to capture the flow patterns of the scrubbing liquid inside the test rig. The camera was able to achieve up to 5000 frames per second at high resolution. LED lights were used to fulfil the high lighting requirement when using a high speed camera. As seen in Figure 1 red food dye was added to the water to assist with visualising the water flow through the scale model.

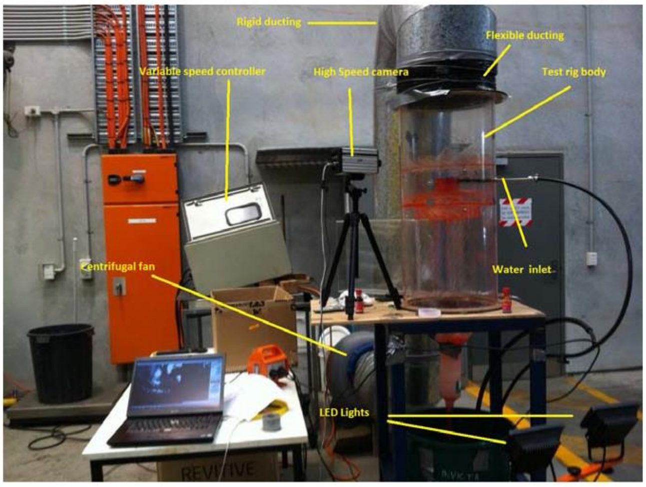


Fig 1 – Experimental test rig

Measurements

Tests were carried out with air drawn through the scrubber scale model at different flow rates without and with water addition. As a starting point the air flow through the scale was chosen so that the air velocities through the scrubber scale model would be similar to the gas velocities through a full scale scrubber module. The other option of achieving full Reynolds number similarity with the flow in a full scale scrubber module would have required air velocities in the scale model approximately ten times those in the full scrubber module which was judged impractical.

Air velocities were measured in traverses across the scrubber scale model below and between the scrubbing vanes without water addition. Pressure drops across the scrubber scale model were also measured for different air flow rates without water addition. These measurements were used for CFD model validation.

Further tests were carried out with water added to the trough on the water distribution cone as air flowed through the scale model. It was not possible to measure the air flow distribution in the main body of the scale model because the water would have damaged the hot wire anemometer but it was still possible to measure velocities in the scale model inlet duct (and therefore the air volumetric flow rate) and the pressure drop across the scale model. The scrubber scale model air flow rate that corresponds to a typical inlet gas flow for a full scale scrubber module was multiplied by the design rate of water addition per unit inlet gas volume flow for a full scale scrubber module to determine the required rate of water addition for these tests. Pressure drops across the scrubber scale model were measured at this fixed water flow rate for different air flows while the motion of the water (with red food dye added) through the scrubber module was tracked with the high speed camera.

CFD modelling details

The flows in the scrubber scale model and two different full scale scrubber modules were simulated using the CFD package ANSYS Fluent. The geometries for the simulations were generated and then imported into ANSYS Workbench platform to be tailored for CFD use. For each of the geometries a hybrid mesh was generated with varied cell sizes and inflation layer thicknesses until a trade-off size was selected that provided the best results with the least cell count. The robust k-ε turbulence model was used to obtain initial solutions and these solutions were refined by iterating further with the Reynolds stress model (RSM) which is widely accepted as being more accurate for highly swirling flows (Weber *et al.*, 1990).

Initial simulations of the scrubber scale model were carried out without water or dust addition to provide data that could be compared with experimental measurements. More detailed multiphase simulations of the scrubber scale model were then carried out. These multiphase simulations treated the water a separate fluid (Eulerian approach) and tracked the dust particles individually (Lagrangian approach). The dust particle size distribution used in the multiphase simulations of the scrubber model was obtained by fitting a Rosin-Rammler curve to a measured bagasse ash particle size distribution and is shown in Figure 2.

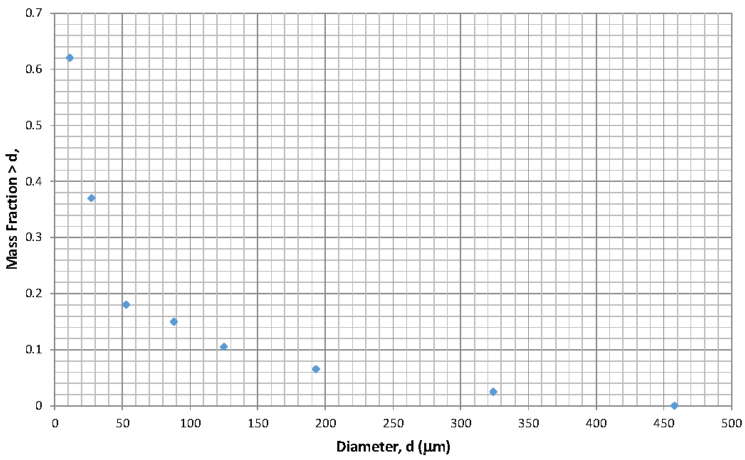


Fig 2 – The cumulative size distribution of dust particles used in the scrubber scale model multiphase simulations.

Results

Figure 3 shows the maximum measured velocity magnitudes between the scale model scrubbing vanes when the total air flow through the scale model was 0.287 kg/s. The pressure drop across the scale model corresponding to these measurements was approximately 380 Pa. The velocity distribution was non uniform with velocity magnitudes varying by a factor of 2.9. The highest velocities are on the inlet duct side of the scale model.

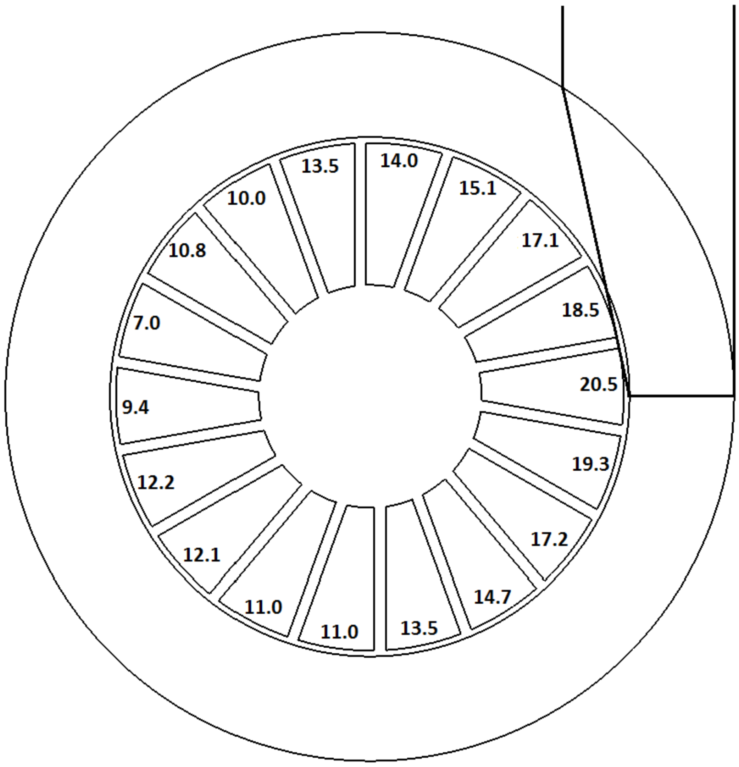


Fig 3 – Measured maximum velocity magnitudes (m/s) between the scale model scrubbing vanes.

Figure 4 and Figure 5 show the measured tangential and axial air velocities in a plane below the scrubbing vanes for traverses parallel to and perpendicular to the direction of the flow in the inlet duct for the same conditions as the measurements in Figure 3. Figure 4 shows that the velocity distribution is not symmetrical with higher velocities on the side of the scale model opposite to the direction of flow in the inlet duct (bottom of the sketch in Figure 4). The minimum velocities are also closer to the side of the scale model opposite the direction of flow in the inlet duct. Velocity measurements along the traverse perpendicular to the direction of flow in the inlet duct also show non symmetrical flow with the highest velocities near the inside wall of the scale model (left of the sketch in Figure 5) and near the protrusion of the inlet duct into the scale model. The measured axial velocities in Figure 4 and Figure 5 show much less variation across the width of the scrubber scale model than the tangential velocities.

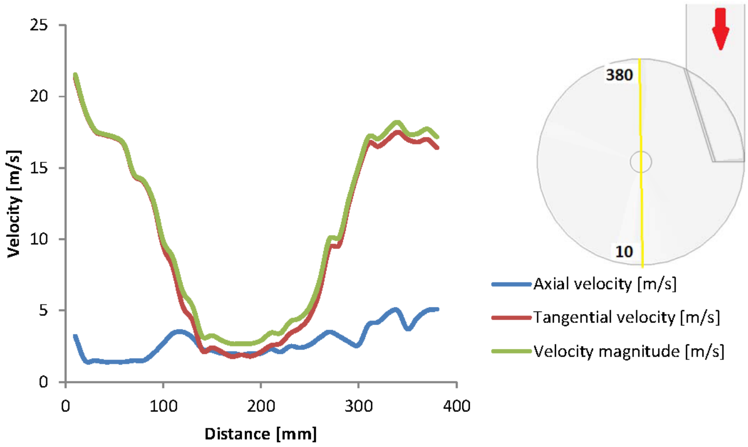


Fig 4 – Measured velocity components and velocity magnitudes (m/s) along a traverse parallel to the longitudinal axis of the scale model inlet duct. Locations of the 10 and 380 mm distances are shown on the sketch.

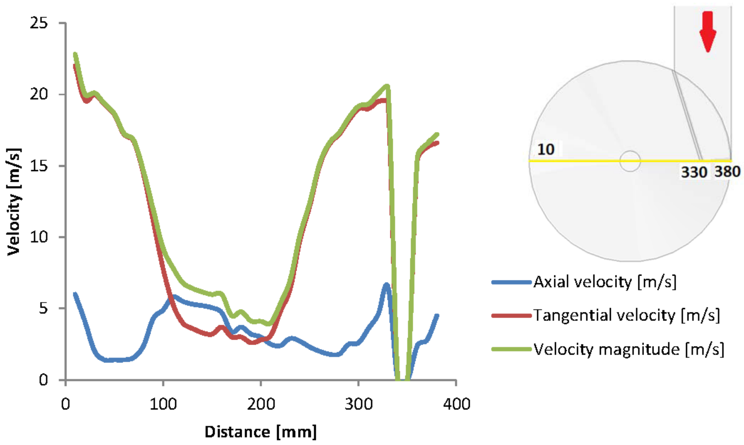


Fig 5 – Measured velocity components and velocity magnitudes (m/s) along a traverse perpendicular to the longitudinal axis of the scale model inlet duct. Locations of the 10, 330 and 380 mm distances are shown on the sketch.

Figure 6 shows a plan view of the predicted air velocity contours through the scrubbing vanes of the scale model. This simulation did not include any water addition. The highest velocities are predicted to be on the inlet duct side of the scale model; this is consistent with the measurements in Figure 3. However the maximum air velocity predictions in the bottom of Figure 6 are higher than the corresponding measurements in Figure 3. The flow in Figure 6 is asymmetrical due to the position of the inlet duct.

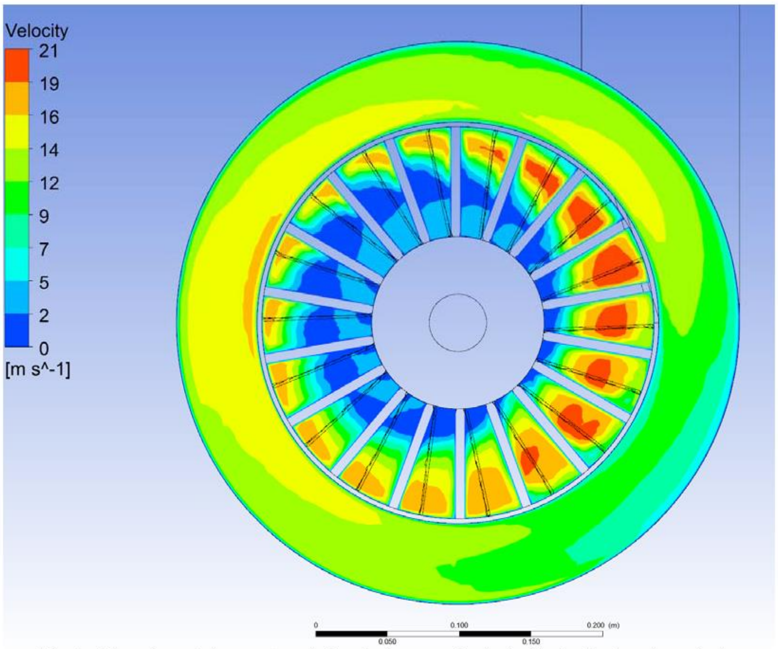


Fig 6 – Plan view of the predicted air velocity magnitude (m/s) distribution through the scrubbing vanes of the scale model.

Figure 7 compares the measured static pressure drops across the scale model for the tests with varying air flow rates with pressure drops predicted with the CFD simulations. The predicted pressure drops are slightly higher than the measured pressure drops but show the same trend.

Further pressure drop measurements were carried out with different air flows and a fixed rate of water addition (0.12 L/s). These measurements are plotted along with the pressure drop measurements without water addition in Figure 8. At low air flows, adding water increased the pressure drop across the scale model by approximately 6%. At high air flows, adding water increased the pressure drop by approximately 30%. Note that in a full scale operating scrubber there will be a reduction in gas volume flow as the gas is cooled by the water; this should partially compensate for the increase in pressure drop due to water addition. This effect will not be picked up by the scale model experiments because air enters at ambient temperature and there will be little cooling of this air when water is added.

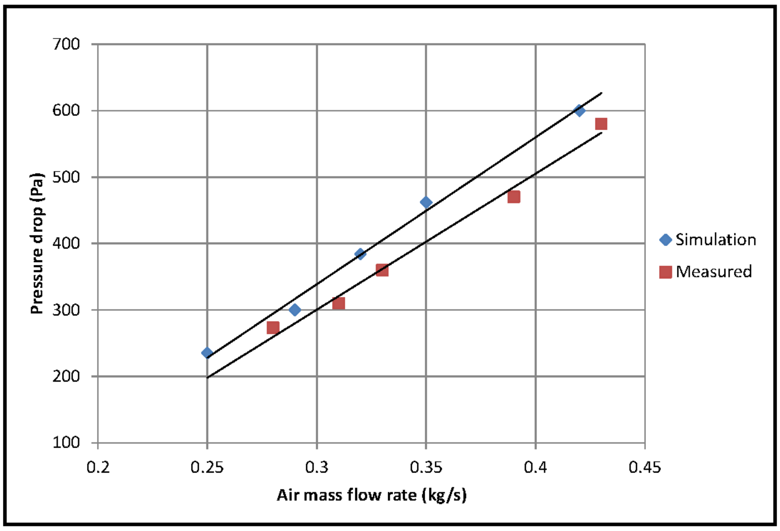


Fig 7 – Predicted (from the CFD simulations) and measured pressure drops (Pa) across the scrubber scale model for different air flow rates (kg/s) without water addition.

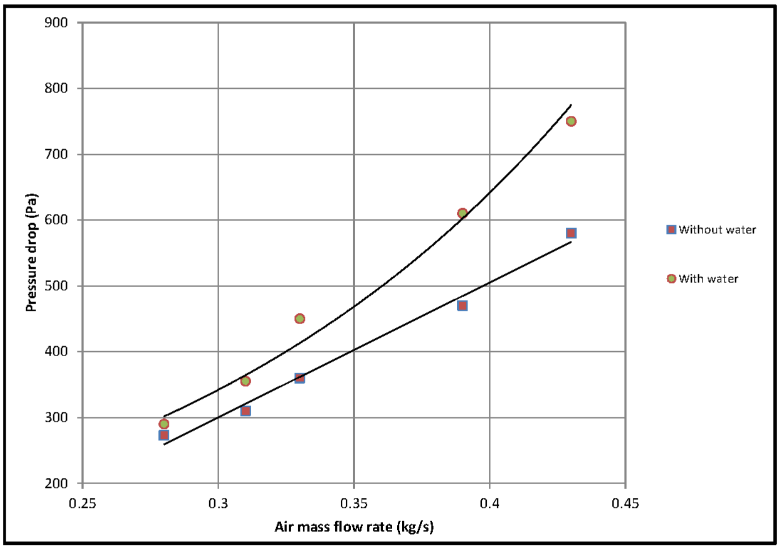


Fig 8 – Measured pressure drops (Pa) across the test rig at different air flow rates without and with water addition.

Figure 9 shows that a highly agitated region of splashing water forms between the scrubbing and demisting vanes of the scale model with water addition when the air velocity exceeds 20 m/s in the region around the water distribution cone. It appears that a lot of the gas cleaning in a full scale scrubber module, the removal of the smaller particle sizes in particular, can occur as it passes through this region. It was found that when the air flow rate was very high, this region of splashing water lifted up into the demister vanes. This is a likely mechanism for the increased droplet carryover experienced by tangential entry fixed vane scrubbers when a boiler is operated with very high gas flows resulting from tramp air leakage and/or the boiler operating at high steam loads as seen in Figure 10. However it is noted that the distance between the scrubbing and demisting vanes is much greater in a full scale scrubber so this water overflow into the demister vanes would be less likely to occur.

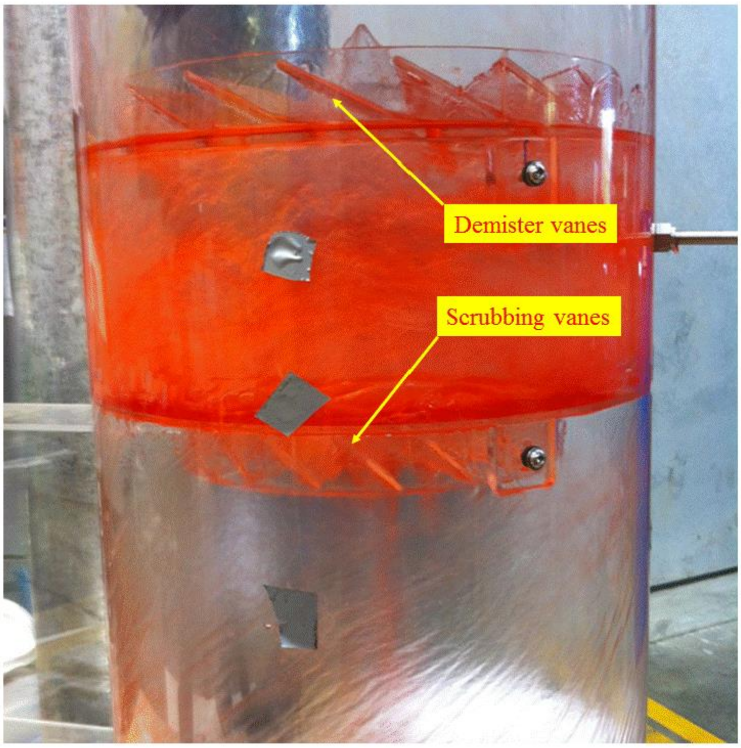


Fig 9 – Part view of the scrubber scale model showing the agitated water region between the scrubbing and demisting vanes.

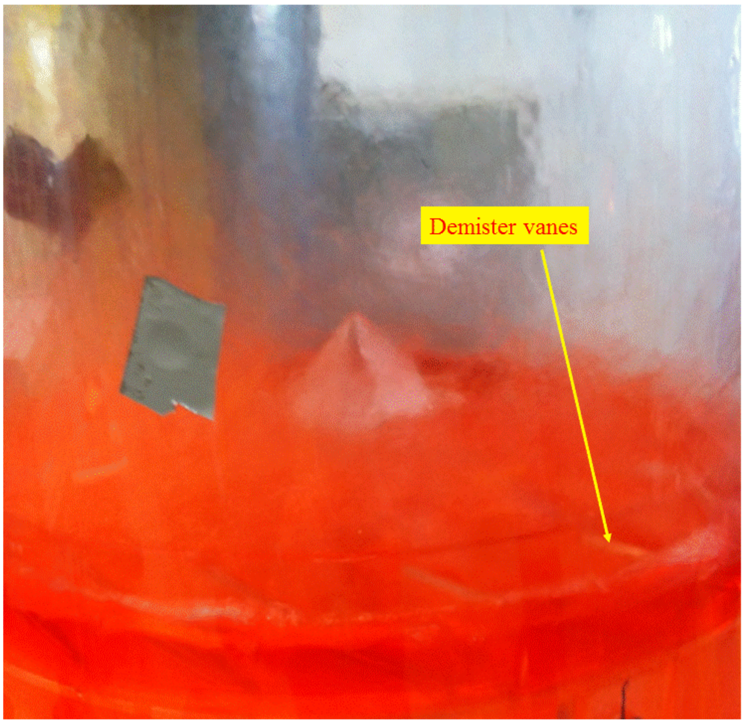


Fig 10 – Part view of the scrubber scale model showing the water overflowing into the region above the demisting vanes when the air flow is very high.

One way to reduce the risk of water overflowing into the demister vanes in a full scale scrubber module would be to increase the distance between the scrubbing and demister vanes. However, unless there is a large gap between the demister vanes and the exit of the scrubber module, the scrubber module would have to be made taller. In most cases this would be impractical. Another option would be to install an annular ‘breakwater’ between the levels of the scrubbing and demisting vanes (Figure 11) to help prevent the region of splashing water rising to the level of the demister vanes. CFD modelling of a full scale scrubber module without and with this modification predicts that provided the inner diameter of the annular ‘breakwater’ is no smaller that the outside diameter of the demister vanes, there will be minimal additional pressure drop across the scrubber module.

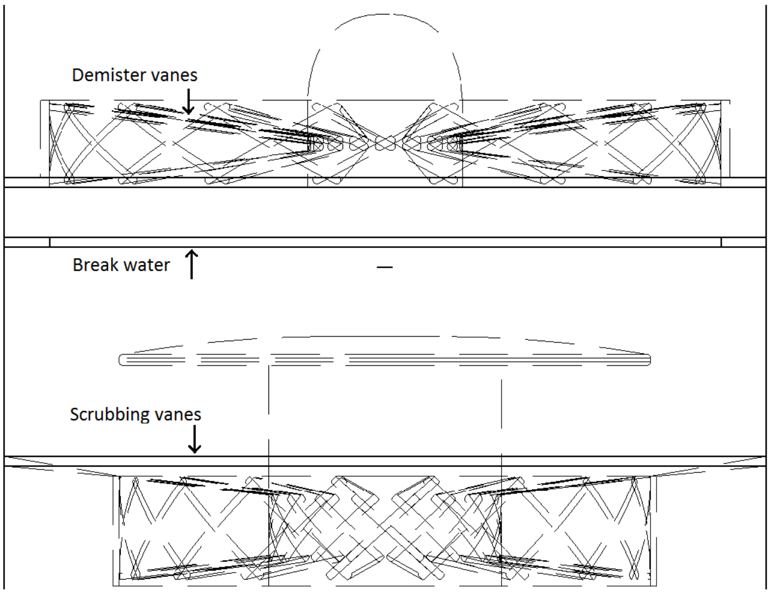


Fig 11 – Part view of the simulated geometry with the annular 'break water' below the demisting vanes.

Figure 12 shows the break-up process of the water curtain that forms from the edge of the water distribution cone captured by the high speed camera. The photographs in this figure were taken with high air flow through the scrubber scale model. As the water leaves the distribution cone it forms sheets; each sheet then deforms into a concave shape (bag) that finally bursts into small ligaments and droplets due to the shear forces between the water and the air.

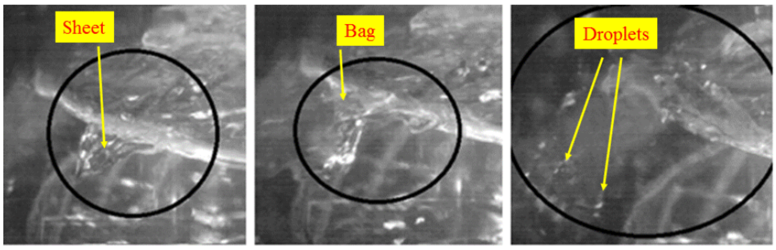


Fig 12 – Photographs showing the break-up of a sheet from the water distribution cone in the scrubber scale model at high air flow.

At lower air flows, the size of the liquid sheets before deformation into concaved bags increases. The maximum bag size before the bag bursts also increases. The resulting ligaments and droplets are likewise larger than those formed at high air flows (Figure 13).

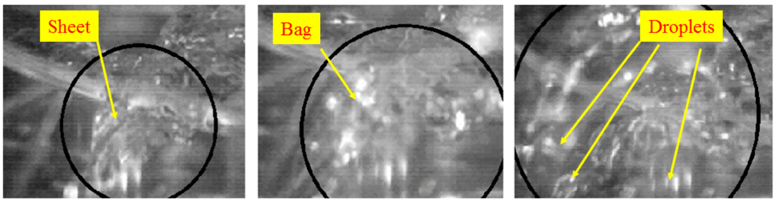


Fig 13 – Photographs showing the break-up of a large sheet from the water distribution cone at low air flow.

Photographs from the high speed camera identified re-entrainment of water droplets from the end of the wall film at the bottom of the scrubbing vanes of the scale model (Figure 14). The high air velocities and velocity gradients in this region help break down the water film into tiny droplets. This maximises the exposed surface area of the water droplets and assists with the collection of the smaller dust particles that, due to their lower inertia, are less likely to be collected on the walls of the scrubber.

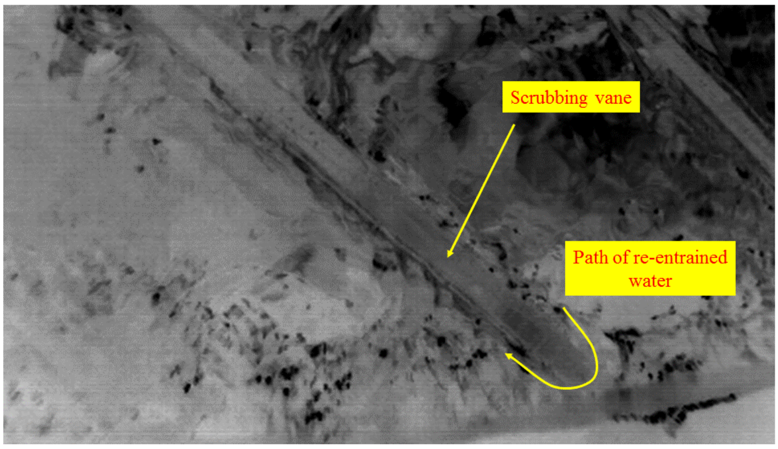


Fig 14 – Re-entrainment of liquid film breaking down into droplets. Water displays as black in the photograph.

Figure 15 shows the predicted trajectories of the ash particles from the multiphase simulations of the scrubber scale model. The larger particles, which are a relatively small proportion of the total ash mass flow (Figure 2), are predicted to hit the inside wall of the scrubber below the scrubbing vanes. These particles have high kinetic energy and are predicted to penetrate the surface of and be collected by the film of water on the inside wall of the scrubber scale model. The smaller diameter particles (shown in blue) are most likely to travel up through the scrubbing and demisting vanes to the outlet.

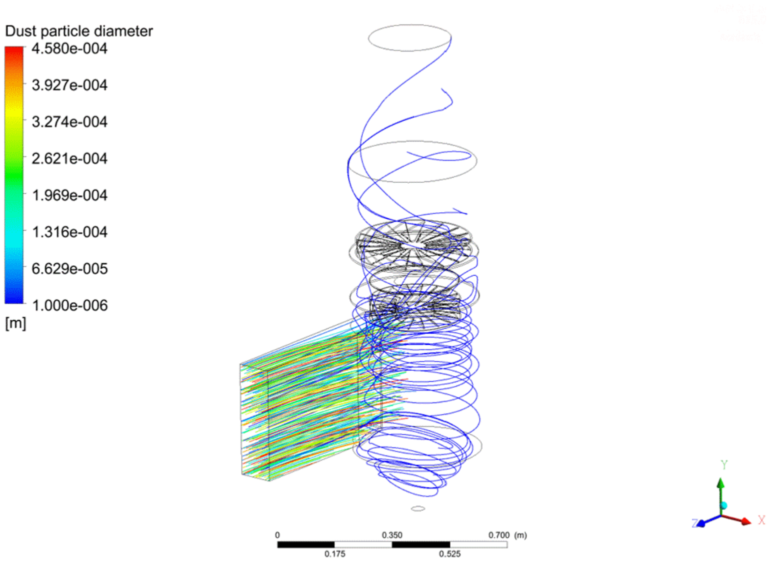


Fig 15 – Predicted ash particle trajectories through the scrubber scale model.

Conclusions

These experimental and modelling investigations of the flow patterns in a scrubber scale model have provided useful insights into what happens in a full scale tangential entry fixed vane wet scrubber.

The tests without water addition and the single phase modelling of the scrubber scale model both found the tangential air velocities to be much higher near the inside of the scrubber walls. There was much less measured variation in the axial air velocity through the scrubber scale model.

The scrubber scale model tests with water addition and varying rates of air addition identified mechanisms for the high droplet carryover that occurs in full scale tangential entry fixed vane scrubbers when the water flow and/or gas flow are too high. A possible corrective measure was proposed with the aid of computer modelling.

The high speed photography identified mechanisms for the water film break-up at different locations in the scrubber scale model.

Acknowledgements

The assistance provided by Neil McKenzie and Barry Hume with the experimental work and the funding support of Sugar Research Australia are gratefully acknowledged. Bingera Mill and Isis Mill are acknowledged for allowing and assisting with scrubber inspections carried out in this project.

REFERENCES

Akiyama T, Marul T (1986) Experimental investigation on dust collection efficiency of straight-through cyclone with air suction by means of secondary rotational air change. *Industrial and Engineering Chemistry Process Design and Development* **25**, 914-918.

Ali M, Yan C, Sun Z, Gu H, Mehboob K (2015) Dust particle removal efficiency of a venturi scrubber. *Annals of Nuclear Energy* **54**, 178-183.

Calvert S (1970) Venturi and other atomizing scrubbers efficiency and pressure drop. *AIChE Journal* **16**, 392-396.

Costa M, Ribeiro A, Tognetti E, Aguiar M, Goncalves J, Coury J (2005) Performance of a venturi scrubber in the removal of fine powder from a confined gas stream. *Materials Research* **18** (2) on-line version.

Dirgo J, Leith D (1985) Cyclone collection efficiency: comparision of experimental results with theoretical predictions. *Aerosol Science and Technology* **4**, 401-415.

Guerra VG, Béttega R, Gonçalves JAS, R. CJ (2012) Pressure drop and liquid distribution in a venturi scrubber: experimental data and CFD simulation. *Industrial and Chemical Engineering Research* **51**, 8049-8060.

Kim J, Lee K (1990) Experimental study of particle collection by small cyclones. *Aerosol Science and Technology* **12**, 1003-1015.

Mann AP, Rasmussen R (2011) Improving the combustion performance of the Farleigh No. 3 boiler. *Proceedings of the Australian Society of Sugar Cane Technologists* **33**, (electronic format) 7 pp.

Pak SI, Chang KS (2006) Performance estimation of a Venturi scrubber using a computational model for capturing dust particles with liquid spray. *Journal of Hazardous Materials* **138**, 560-573.

Plaza F, Dixon TF, Dickenson NL, Fitzmaurice AL, Owens M (1999) Performance of baffled boilers with redesigned convection banks. *Proceedings of the Australian Society of Sugar Cane Technologists* **21**, 432-437.

Shin MS, Kim HS, Jang DS, Chung JD, Bohnet M (2005) A numerical and experimental study on a high efficiency cyclone dust separator for high temperature and pressurized environments. *Applied Thermal Engineering* **25**, 1821-1835.

Weber R, Visser BM, Boysan F (1990) Assessment of turbulence modeling for engineering prediction of swirling vortices in the near burner zone. *International Journal of Heat and Fluid Flow* **11**, 225-235.