NUMERICAL AND EXPERIMENTAL STUDY OF THE FLOW IN VACUUM PANS

By

L.F. ECHEVERRI¹, P.W. REIN¹ and S. ACHARYA²

¹Audubon Sugar Institute, LSU AgCenter, Louisiana, USA
²Mechanical Engineering Dept. LSU, Louisiana, USA

Email: PRein@agcenter.lsu.edu

KEYWORDS: Crystallisation, Vacuum Pans, Circulation, CFD.

Abstract

The design of vacuum pans has evolved empirically in an attempt to achieve satisfactory circulation and heat transfer rates, variables that are critical for the efficient crystallisation of sugar. Computational fluid dynamics (CFD) and modern velocimetry techniques have been applied to the analysis of fluid flow in vacuum pans, in an attempt to understand the flow patterns and to identify alternatives to enhance the performance of pans by improving circulation. To analyse the flow inside vacuum pans, a multiphase CFD model is applied, which should incorporate aspects such as the non-Newtonian behaviour of massecuites, the presence of a free surface, and the gravity buoyancy forces that drive the flow. These conditions make the flow in vacuum pans particularly complex to analyse, and highly demanding from a computational point of view. To verify the CFD results, an experimental scaled test rig was designed and constructed to represent the major features of fluid flow in full-scale vacuum pans. Laser anemometry techniques have been applied to measure the fluid velocity field under different conditions and in evaluating the effect of variables such as evaporation rate and strike height. This paper presents some numerical results and experimental data obtained from the test rig. The methodology used in the modelling has been extended to the analysis of full-scale batch and continuous vacuum pans. This has enabled a prediction of the circulation rate as a function of factors like the shape of the pans, the calandria design, and operational variables such as the massecuite boiling height and massecuite viscosity.

Introduction

The design of vacuum pans has evolved through an empirical process because of a lack of understanding of the fundamental heat transfer and fluid flow phenomena and the intrinsic difficulty in performing useful field measurements. Limited information is available from experiences in full-scale pans using radioisotope tracers (Wright, 1966) and hot wire anemometry techniques (Bosworth, 1959; Rackemann and Stephens, 2002). A correlation was obtained experimentally by Rouillard (1985) for evaporation as a function of the calandria tubes geometry, vacuum, heating vapour pressure, massecuite properties and head. In recent years, a few studies have reported the use of computational fluid dynamics (CFD) in the analysis of vacuum pans (Bunton, 1981; Stephens, 2001; Rein et al., 2004), revealing the complexity of the problem.

In general, it is believed that, during the boiling process, the vapour bubbles generated agitate their surroundings, mixing the hot liquid close to the exchange surface with the rest of the stream, and triggering high heat transfer coefficients typical of boiling regimes. However, boiling is still far from being fully understood because of its complex nature and the uncertainty regarding interaction between the factors involved. Particularly for vacuum pans, theories including nucleation, coalescence of bubbles, formation of slugs or columns of vapour, temperature differences, flashing, eruptions, etc. have been proposed to describe the process, but the lack of information has not enabled reasonable conclusions to be drawn. However, it is clear that the circulation should be as high as practically possible for quality, recovery and capacity reasons (Rein et al., 2004), and that convective boiling plays a vital role in vacuum pans, with the
density difference between the vapour phase and the massecuite providing the driving force for massecuite movement. The balance between buoyancy forces and frictional resistance determines the circulation (Webre, 1959) and, as normal in convection regimes, the heat transfer is strongly interrelated with the fluid velocity or circulation.

Research into the flow within vacuum pans applying CFD and modern anemometry techniques is being carried out at Louisiana State University. The main goal is to identify alternatives to enhance the performance of vacuum pans by improving the circulation. This paper presents some initial experimental and computational results.

Materials and methods

Experimental work

The fluid flow in vacuum pans is studied experimentally using particle image velocimetry (PIV) to determine the flow field in a lab-scale test rig designed to represent circulation in vacuum pans and to provide data to verify CFD simulations (Figure 1). The test rig geometry is based on an A-pan design that provides satisfactory performance.

To emulate the process in a simplified and controllable manner, spargers are used to inject air into vertical channels corresponding to the calandria tubes. The injected air represents the vapour generated, and the buoyancy resulting from density differences between the liquid and gas phase produces the circulation driving force. The test rig is constructed with flat plexiglas walls, which allow the passage of laser light used by the PIV.

PIV is a modern anemometry technique that measures the velocity fields in an illuminated plane. The technique uses a pulsed laser light sheet synchronised with a high-resolution digital camera (charged couple device – CCD) that captures images of tracer particles in the fluid. The flow is seeded using micro spheres that scatter the laser light, allowing recording of their position in images acquired with the CCD during each laser pulse. The tracer particles must have the right density and size, being small enough to accurately follow the flow, and large enough to be detectable by the CCD.

Image pairs are acquired using a very small separation time (e.g. 350 µs), which is set to measure accurately the particle displacement and hence the velocity. For the analysis, the images are subdivided into small interrogation areas, and cross correlation is used to compare each pair of images using a Fast Fourier Transform, finding the average displacement for each area. PIV measurements give an expected accuracy of 1% of the full scale velocity (Mercer, 2003).

The PIV system uses a 3 mm thick laser sheet generated by two 15 Hz New Wave Gemini Nd:YAG 532 nm lasers (Neodymium doped Yttrium Aluminium Garnet). A 30 Hz Kodak Megaplus ES 1.0 CCD camera, with resolution 1008 x 1012, and provided with an optical zoom AF Nikkor 50mm f/1.8D, acquires images of the plane illuminated by the laser sheet, where the position of 10 µm diameter silver coated glass spheres used as tracer particles is recorded. The PIV is operated in double exposure mode, and for the analysis a cross correlation with a 42 x 42 pixel interrogation area is used. During 100 seconds, 400 image pairs are acquired, and instantaneous velocity results generated from each image pair are averaged to determine the velocity field at each position.

![Fig. 1a—Test rig design.](image-url)
The experiments are conducted at room temperature, using tap water initially to represent the liquid phase (998 kg/m$^3$, 0.001 Pa.s), which conveniently allows benchmarking against experimental and analytical experiences reported on the rise of air bubbles in water. Liquid heads between 0–300 mm with respect to the top of the channels are tested, while air is injected at a rate between 2831. and 3775 cm$^3$/s. The size of the bubbles is determined by image analysis, calculating an equivalent diameter in the case of non-spherical bubbles.

**Computational simulations**

The fluid flow in the test rig and in vacuum pans is simulated using a commercial CFD code (Fluent, 2003). An Eulerian-Eulerian multiphase approach is applied, which considers that the secondary phase can be treated as a continuum and solves the Navier-Stokes equations simultaneously for each phase. Although strictly three phases are present during sugar crystallisation (mother liquid + vapour + sugar crystals), the high viscosity of massecuite and relatively small size of sugar crystals make it reasonable to assume that the liquid and solid phases move together, so a two-phase flow model is used (vapour and massecuite). The exchange of momentum or interaction between the phases is an important aspect in multiphase flows, and in the Eulerian-Eulerian approach plays its role coupling the momentum equations that are being solved for each phase.

For the calculation of the momentum exchange, empirical or semi-empirical models are normally used. In the cases studied, the secondary phase (air or vapour) separates quickly from the liquid at the free surface, and becomes the primary phase above. Accordingly, for the numerical analysis, an exchange model appropriate for situations where the secondary phase becomes the primary phase in part of the domain is used.

The gas phase is injected in the tubes using mass source terms corresponding to typical evaporation rates in vacuum pans. The primary buoyancy is generated by the difference in the liquid and vapour density. The buoyancy forces caused by liquid temperature differences are assumed negligible with respect to those caused by density difference between the phases; thus the system can be treated as isothermal, avoiding the solution of the energy equation.

**Results and discussion**

**Test rig**

The experimental results obtained with the test rig may be classified in two sections:

1. **Single-phase section**: In the area of the test rig corresponding to the bottom of the pan, only few and small bubbles are present, indicating a single-phase liquid region. Good PIV results are relatively easy to obtain in this region, clearly detecting the seeding particles. The minimal presence of gas phase in the bottom region makes it possible to integrate the liquid flow as a measure of circulation.

2. **Two-phase section**: A significant quantity of gas bubbles is present in the rectangular channels corresponding to the calandria tubes, in the region above the top calandria plate, and in the downtake, where many bubbles are dragged down and reduce the downtake effective area. The bubbles scatter light that is captured by the CCD, making it difficult to distinguish the seeding particles from the numerous bubble contours that are recorded. Therefore, the PIV results obtained for this region do not correspond to the liquid phase, but mainly to the gas phase.
In spite of using equal diameter orifices in the air spargers, some dispersion in the size of the bubbles is observable, probably as a consequence of the different flow conditions between the channels, the coalescence and the collapse of bubbles that cross the free surface. The smaller bubbles (diameter < 3 mm) are approximately spherical, while the larger bubbles tend to have an elliptical shape. An average equivalent diameter of 4.44 mm is found, rounded to 4.5 mm for analysis purposes. The measurements show a rise velocity of the gas phase between 0.23–0.26 m/s, and a slip velocity around 0.10 m/s, which are consistent with reported values of the rise velocity of 4.5 mm air bubbles in water (dirty water 0.17 m/s, clean water 0.25 m/s, Patro et al., 2001).

**Effect of the liquid head on circulation**

The circulation rate is a key performance metric of interest. The PIV velocity measurements in the single phase region are used to calculate this circulation rate. Results obtained for steady airflow (corresponding to a constant and uniform evaporation rate) are shown in Figure 2 and suggest initially that, for low liquid heights (<100 mm), a circulation rate proportional to the head is obtained. After a critical height is passed, the circulation remains almost constant, independent of the head (Figure 2).

![Fig. 2—Circulation rate vs. liquid head.](Image)

In vacuum pans, for well-established reasons, the evaporation rate decreases during the strike, causing the circulation to decrease correspondingly. Clearly, the maximum evaporation occurs at the start of the strike, but the perception that the maximum circulation takes place simultaneously is not supported by the results obtained, which show that for low heads a ‘bottle-neck’ effect above the calandria is obtained, restricting the liquid from flowing toward the downtake. This behaviour was inferred years ago by Allan (1962), who proposed that a lower head means a smaller sectional area, and therefore an increase in resistance and less circulation. This result points to the existence in the design of continuous vacuum pans of an optimum massecuite height.

**Effect of evaporation rate on circulation**

It is acknowledged that, in vacuum pans, the evaporation rate and massecuite circulation are interrelated. Surprisingly, the test rig data did not show a significant effect of injected airflow on the circulation rate within the range of conditions evaluated (Figure 2). This suggests that higher evaporation does not mean automatically higher circulation above a given vapour generation rate.

**Flow patterns**

In the bottom region, the results show smooth flow patterns for low circulation rates (Figure 3a); while, for higher circulation, a flow separation below the inner bottom corner of the calandria is developed (Figure 3b). In the two-phase region, large-scale separation and vorticity are consistently observed at the side wall above the calandria and in the downtake.

**Comparison between experimental and computational results**

A comparison between the experimental and computational results shows that the CFD simulations overestimate the flow (Figure 2), predicting higher bubble rise velocities and liquid circulation than measured, by a factor of between 1.7 and 2. From a qualitative point of view, similarities in the flow patterns are identifiable, with the CFD liquid phase predictions agreeing with the PIV results in the bottom
single-phase region (Figure 3), and the CFD gas phase predictions qualitatively matching with the PIV results in the two-phase flow section. For the high head / high circulation case, the large vortex observed in the bottom region is the reason for the low velocities within the channels located closer to the downtake; these results also appear in the CFD predictions. Under these conditions, the experimental and numerical results show lower circulation through the channels closer to the downtake.

Fig. 3a—Low liquid head (without air 25 mm). Airflow rate 2831 cm³/s. Velocity vectors measured with PIV (left) and liquid velocity vectors predicted with CFD (right).

Fig. 3b—High liquid head (without air 280 mm). Airflow 2831 cm³/s. Velocity vectors measured with PIV (left) and liquid velocity vectors predicted with CFD (right).
Since CFD overprediction of the gas rise velocity and circulation have been observed, it is believed that the drag forces are being underestimated during the simulations. Two possible causes that are being investigated are:

1. **Drag coefficient**

To characterise the dynamics of gas dispersed in liquids, a drag coefficient is used. There are many factors that influence the behaviour of rising bubbles, such as size, temperature, shape, rising pattern, age, power law index of the liquid, purity, contamination, surface tension or presence of surfactants.

Many drag coefficient correlations and attempts to correlate different experimental data have been reported; observing a high dispersion and significant differences between ideal systems (spherical bubbles, pure liquid) and non-pure systems (distorted bubbles, impurities present), where the bubble rising velocity can be only half as much as under ideal conditions.

The interaction model used in the CFD computations utilises a drag coefficient expression developed by Schiller and Naumann (1933). The drag coefficient correlations have been developed mostly from experiments of single isolated bubbles rising in stagnant liquid columns, or for low void fractions (<3%).

These drag coefficients are used normally in the development of multiphase models for CFD codes, but should be strictly applicable to the analysis of dilute systems (Behzadi et al., 2004).

The balance between drag and buoyancy forces changes as the void fraction increases; this is caused by the proximity between the bubbles favouring interaction forces, or by the reduction of buoyancy forces as a consequence of lower density in the surrounding gas-liquid mixture (Rusche and Issa, 2000).

2. **Turbulence**

Simulations of bubbly flows presented in the literature include fully turbulent models, even when a low void fraction is displayed. The mixing of fluid particles caused by turbulence results in a rapid rate of diffusion of momentum and heat, leading to increased heat transfer, drag and pressure drop. In most situations, bubble motion is characterised by randomness, inducing fluctuations in the liquid phase flow and making turbulence an inherent part of bubbly flows.

Experiments have demonstrated that turbulence increases with the void fraction, which has a considerable effect on the drag force, even for low void fraction flows (Behzadi et al., 2004). The fluctuating velocities of the bubbles and the liquid phase are higher as the void fraction is increased (Bröder and Sommerfeld, 2002), while the rising velocity of the bubbles is drastically reduced as the turbulence intensity increases (Rusche and Issa, 2000).

Unfortunately, the effect of turbulence in multiphase flows is not yet completely understood. Note a laminar flow model is used for these predictions.

**CFD simulation of vacuum pans**

Some initial CFD results obtained in the modelling of full-scale vacuum pans are presented. The conditions in the pan are chosen to approximate the last stage of a B massecuite boiling (Table 1). It is acknowledged that the CFD strategy still requires improvement and no definite quantitative conclusions can yet be drawn.

Overprediction of the flow velocities is expected, but even so the flow patterns are expected to be qualitatively realistic and helpful in understanding the phenomena inside the vacuum pans. In addition, it should be borne in mind that, in practice, circulation will affect evaporation rate and hence vapour generation, probably accentuating the effects found here.

**Batch vacuum pans**

In this analysis, cylindrical polar coordinates are used defining an axisymmetric case. In general, the CFD results suggest the presence of a vorticity near the side wall above the calandria (Figure 4a). This is accentuated in the case of flared pans and explains the poor performance experienced with this design (Rein et al., 2004). A larger vorticity appears over the top calandria plate towards the downtake, which in the case of high circulation rates, extends inside the downtake, reducing its effective area.

The predicted velocities of massecuite in the bottom are low (Figure 4a), and so the resultant pressure distribution resembles a hydrostatic pressure field (Figure 4b). The results have shown similar liquid flux rates across the rings that represent the calandria, suggesting that a W-bottom shape can promote without difficulty an even distribution of the massecuite in all the tubes.

Above the top calandria plate, the vapour phase flows accordingly with the liquid phase flow patterns, and shows that some bubbles are dragged down into the downtake (Figure 4b).
With respect to the effect of viscosity and evaporation rate, the simulations suggest that if the massecuite viscosity is doubled (from 20 to 40 Pa.s), the circulation would be reduced by 40% (Table 1). Reducing the evaporation rate by half (from 6.2 to 3.1 kg/h.m$^2$) would decrease the circulation 25%.

As expected, low viscosities and high evaporation rates favour the circulation of massecuite, and therefore the performance of vacuum pans. For the same evaporation rate, a higher viscosity is cause of a higher gas hold-up, higher rise of the free surface, and a larger void fraction in the calandria tubes.

Previous CFD analyses have considered the massecuite as a Newtonian fluid. Massecuites are often reported to be pseudo-plastic non-Newtonian fluids, with a power law index around 0.8. Work on the rise velocity of bubbles in non-Newtonian liquids has shown that the drag coefficient depends on the flow behaviour index of the power law (Dziubinski et al., 2003).

Therefore, in order to obtain accurate CFD predictions, the applied drag model may well need to consider the non-Newtonian index of the massecuite, in addition to the high vapour fraction.
Table 1—Effect of the massecuite viscosity and evaporation rate.

<table>
<thead>
<tr>
<th></th>
<th>Reference case</th>
<th>Higher visc. (1:2)</th>
<th>Lower evap. (2:1)</th>
<th>Non-Newtonian</th>
</tr>
</thead>
<tbody>
<tr>
<td>Massecuite viscosity (Pa.s)</td>
<td>20</td>
<td>40</td>
<td>20</td>
<td>k=20, n=0.8</td>
</tr>
<tr>
<td>Evaporation coeff. (kg/h.m²)</td>
<td>6.2</td>
<td>6.2</td>
<td>3.1</td>
<td>6.2</td>
</tr>
<tr>
<td>Circulation time (s)</td>
<td>39.11</td>
<td>62.66</td>
<td>52.40</td>
<td>33.11</td>
</tr>
<tr>
<td>Circulation rate (m³/s)</td>
<td>2.30</td>
<td>1.44</td>
<td>1.72</td>
<td>2.72</td>
</tr>
<tr>
<td>Ratio</td>
<td>1.00</td>
<td>0.62</td>
<td>0.75</td>
<td>1.18</td>
</tr>
<tr>
<td>Tube outlet void fraction</td>
<td>27.6%</td>
<td>40.4%</td>
<td>15.7%</td>
<td>23.8%</td>
</tr>
</tbody>
</table>

Massecuite density 1470 kg/m³; Massecuite level above calandria 1.2 m.

Circulation time (seconds) = Volume of massecuite in pan/circulation rate.

The inclusion of the viscosity power law in the CFD analysis, using the same previous drag model, a consistency index of 20 kg.s⁻²/m and a power law index 0.8, gives a flow field slightly different from the results obtained assuming Newtonian behaviour (Figure 5). A higher circulation rate is obtained with respect to the Newtonian case (Table 1), this due to computed shear rates >1 s⁻¹ in a large portion of the domain (53%). High shear rates appear in critical regions of the flow, like near the tube walls, where the computed local viscosity is lower than the previously assumed 20 Pa.s. However, if the circulation and shear rates were lower, the opposite behaviour would be possible.

![Fig. 5—CFD results obtained applying the viscosity power law (k = 20, n = 0.8).](image)

As an isothermal flow is assumed for the simulations, the effect of the temperature in the viscosity of the massecuites is not considered. However, near the walls of the calandria tubes, a temperature slightly lower than the heating vapour temperature is expected, which would reduce the viscosity near the walls, and thus lower the frictional resistance.

Continuous vacuum pans (CVPs)

The same CFD strategy, switching the coordinates system from polar to rectangular and adjusting the mass source terms, can be used to analyse CVPs. CFD simulations were performed based on the geometry presented in the literature for the most well known CVPs (van der Poel et al., 1998; Moor, 2001). The same massecuite properties used previously for batch pans are assumed, while the evaporation coefficient is chosen to be 10 kg/h.m² which is based on results obtained for B-strike CVPs (Rein and Msimanga, 1999). In general, the results show a vorticity over the top calandria plate near the downtake similar to the one found for batch pans (Figure 6). The simulations predict a circulation time between 28—32 s, so the massecuite enters the calandria tubes at 0.21—0.24 m/s, while at the tube's exit the void fraction reaches 57—60%.
Fig. 6—CFD prediction of void fraction and massecuite velocity vectors for different CVP designs.

For CVPs, the footing volume is not an issue, and no compromise between bottom volume and circulation resistance exists, so the design can be optimised via larger downtake and bottom volumes. It is believed that a proper CFD analysis will be valuable for improving the design of CVPs and enabling a better understanding of the process inside the vessels.
Conclusion

The circulation of massecuite in vacuum pans has been studied using a lab-scale test rig and a modern anemometry technique, identifying the main flow patterns and assessing the effect of the liquid head and the gas flow rate on the circulation.

The comparison of the experimental result against numerical simulations indicates that the current CFD modelling strategy over predicts the flow velocities, and two possible causes for the difference are identified.

The CFD has been applied in the modelling of batch and continuous vacuum pans, illustrating the effect of some variables on the circulation. CFD modelling offers a tool for progressing efforts to improve the design and operation of vacuum pans.

REFERENCES


LA CIRCULATION DANS LES CUITES: UNE ÉTUDE
EXPÉRIMENTALE ET NUMÉRIQUE

L.F. ECHEVERRI\(^1\), P.W. REIN\(^1\) et S. ACHARYA\(^2\)
\(^1\)Audubon Sugar Institute, LSU AgCenter, Louisiana, USA
\(^2\)Mechanical Engineering Dept. LSU, Louisiana, USA

Email:PRein@agcenter.lsu.edu

MOTS CLEFS: Cristallisation, Cuites, Circulation, CFD.

Résumé
LA CONCEPTION des cuites a évolué d’une façon empirique pour obtenir des résultats satisfaisants en termes de la circulation et du transfert de chaleur, deux variables importants pour la cristallisation. Le logiciel « Computational fluid dynamics, CFD » et des techniques velocimétriques ont été utilisées pour étudier et améliorer la circulation dans les cuites. On s’est servi d’un model CFD qui comprend les aspects non-Newtoniens des massecuites, les effets des surfaces, et les forces qui affectent la circulation. Ces conditions rendent l’analyse particulièrement compliquée. On s’est servi d’un système à l’échelle pilote pour vérifier les résultats. Des techniques d’menomtrye lasag ont été utilisées pour mesurer les vélocités et pour évaluer les effets de variables comme l’évaporation et le volume de massecuite dans la cuite. Ce papier présente les résultats obtenus à l’échelle pilote; on s’est servi de la même technique pour étudier les cuites continues et discontinues. Cela nous a permis de prédire la circulation en fonction de facteurs tels que la forme de l’appareil et de la calandre, la viscosité et le volume de la massecuite.

NUMERICAL AND EXPERIMENTAL STUDY OF
THE FLOW IN VACUUM PANS

L.F. ECHEVERRI\(^1\), P.W. REIN\(^1\) y S. ACHARYA\(^2\)
\(^1\)Audubon Sugar Institute, LSU AgCenter, Louisiana, USA
\(^2\)Mechanical Engineering Dept. LSU, Louisiana, USA

Email:PRein@agcenter.lsu.edu

Resumen
El diseño de tachos ha evolucionado empíricamente en búsqueda de mejor circulación de la masa cocida y transferencia de calor, variables que son críticas para cristalizar eficientemente el azúcar. Dinámica de Fluidos Computacional (CFD) y técnicas modernas de anemometría láser han sido aplicadas en el análisis del flujo en tachos buscando identificar alternativas para mejorar el desempeño de los tachos mediante el incremento de la circulación. Para el análisis del flujo en los tachos se aplica una simulación CFD que considera condiciones propias del proceso, tales como el carácter múltifase, el comportamiento no-Newtoniano de las masas cocidas, la superficie libre y las fuerzas de flotación debidas a diferencias de densidad. Estas condiciones hacen del flujo en los tachos un caso complejo de analizar y costoso desde el punto de vista computacional. Para verificar los resultados CFD un modelo a escala que representa las principales características del flujo en equipos industriales ha sido diseñado y construido. Técnicas de anemometría se han aplicado para determinar el flujo bajo diferentes condiciones, buscando evaluar el efecto de variables como la tasa de evaporación y la altura de la templa en la circulación. Este artículo presenta algunos resultados computacionales y experimentales obtenidos con el modelamiento a escala. La metodología desarrollada ha sido empleada para el análisis de tachos tipos discontinuos y continuos, permitiendo predecir la circulación de acuerdo a factores como la geometría del tacho, el diseño de la calandria, y variables operacionales como la altura y viscosidad de la masa cocida.