Third International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
10-12 December 2003

APPLICATION OF CFD IN THE SUGAR INDUSTRY

Terry DIXON, Anthony MANN, Phil HOBSON, Floren PLAZA, Steve PENNISI and Rod STEINDL

Sugar Research Institute, Box 5611, Mackay MC 4741, AUSTRALIA

ABSTRACT

The paper summarises a wide range of ongoing applications of CFD in the sugar industry. Principal among these have been the development of bagasse combustion technologies and understanding the processes involved in bagasse combustion and steam generation. SRI has been active for over 10 years in CFD with the FURNACE code, encompassing the full range of applications from fundamental code development, the generation and commercialisation of new ideas and technologies, to the resolution of practical plant problems. Typical areas of application have included particle erosion, corrosion and heat transfer in tube bundle units, particle drying, ignition and burnout dynamics, the development of advanced boiler designs and more recently CO and NO_x pollutant generation and reduction.

CFD has been applied extensively in the design of the new-generation SRI clarifier that has achieved throughput increases of over 75% with improved clarified juice quality. Other applications of CFD that are summarised include the modeling and design upgrading of evaporators, capacity and design improvements including stirrer retrofits to vacuum pans used in sugar crystallization, and the development of bagasse gasification technology for advanced power generation.

It is demonstrated that the full benefits of CFD in the delivery of commercial outcomes, for new technologies and the solution of operating plant problems, are achieved through close interaction between code development and validation via full-scale plant simulation. This two-way interaction enhances the code fundamentals by focusing on practical issues and increases the confidence of end-users in the capabilities and accuracy of the CFD predictions, to an extent that justifies firm engineering decisions on commercial plant based solely on the characteristics predicted by the code.

Keywords: Combustion, CFD, Bagasse, Evaporation, Crystallisation, Clarification, Gasification.

INTRODUCTION

The sugar industry in Australia has an evolving and developmental culture that strives to advance its wide range of technologies to remain internationally competitive. In recent years particular emphasis has been directed to the areas of bagasse combustion, steam generation, sugar juice clarification and juice boiling and crystallization. Combustion and steam generation command by far the single largest investment in capital plant for sugar manufacture. The operating and maintenance costs associated with steam generation also demand attention to increase performance and profitability. Since 2002 CFD has also been applied to advanced technology development in the gasification field.

SRI has maintained for over 15 years an integrated program of research and development, full-scale plant demonstration and commercial proving of a range of plant improvements and advanced technologies in the combustion and steam generation field. This activity has been expanded more recently into other areas of sugar manufacture that include juice clarification, boiling and crystallisation. It has emerged that a critical component of any development sequence has been the application of CFD modeling. SRI's involvement with CFD specifically for combustion and boiler applications has embraced the full range of activities from fundamental research into components processes and CFD code development, through to commercial engineering design and problem solving on operational plant. The CFD code (FURNACE) has undergone a sequence of progressive enhancements as the focus of application has shifted.

CFD modeling has established itself as a critical tool for the development of new ideas and advanced technologies. It has been SRI's experience that CFD modeling is capable of predicting qualitative information (trends), and in many cases quantitative information, to within sufficient accuracy to justify engineering design changes on commercial plant across the whole sugar manufacturing spectrum.

The focus of this paper is the applications aspects of CFD modeling in the sugar industry. The modeled systems that are described are highly complex and thus model 'validation' is typically in the form of agreement with qualitative, historical knowledge obtained from plant operators and supervisory personnel. Where available, comparisons with quantitative plant data are included and used extensively. Overall, the modeling has been shown to have good agreement with qualitative trends and acceptable agreement with quantitative data.

BASIS OF CFD MODELS USED AT SRI

SRI's workhorse for CFD modeling in the combustion and boiler fields is the FURNACE code originally developed for pulverized coal fired boilers at the University of Sydney [Boyd and Kent, 1986]. Use is also made of the CFX code for specific applications where an unstructured grid configuration is required for adequate problem representation.

FURNACE is a three dimensional, structured grid CFD code that can predict the standard gas and particle flow patterns, combustion and radiation in utility and industrial boilers firing a range of fuels. The code also has time stepping capability. The code has been modified over a number of years to model the flow and combustion processes in bagasse fired boilers [Luo, 1993; Woodfield *et al*., 1998; Mann *et al*., 2001]. Details of a more recent version of the FURNACE code are described elsewhere [Woodfield, 2001; Mann, 2003]. Some comments are pertinent. The standard k-e, turbulence model is used for all simulations. Lagrangian particle tracking is used to predict the motion of particles (bagasse and ash) through the boiler. Turbulent dispersion of the particles is modelled using the technique of Gasman and Ioannides (1981). The code has been modified to take into account the shape and aerodynamic behaviour of bagasse particles. Bagasse combustion is modelled as a four-stage process: particle heat-up, particle drying, devolatilisation and char burnout.

Several novel features of the code have been developed in recent years in an ongoing enhancement process with John Kent of The University of Sydney. These include the dynamic simulation of the pile burning characteristics of bagasse that has deposited out of suspension on the boiler grate [Woodfield, 2001; Woodfield *et al*., 1999], gas turbine exhaust injection in combined cycle configuration (unpublished work), the simulation of tube erosion processes in individual tubes of heat transfer tube bundles including the incorporation of porous regions [Mann *et al*., 2001; Novozhilov *et al*., 2001], and the accurate simulation of global convective heat transfer within tube bundles, incorporating empirical correlations [Zhukauskas and Ulinskas, 1988], for the prediction of heat transfer coefficients within porous regions that are used to represent the flow through tube bundles [Mann *et al*., 2003]. This takes into account the pressure and temperature drop and flow area restriction caused by the presence of the tube banks. For tube erosion work, sub-grid modelling capability has been added to the FURNACE code. This overcomes the computer processing and storage limitations that prevent detailed modelling of the flow patterns around individual tubes. To use this approach, a FURNACE simulation of the whole boiler is performed but with the inlet conditions (particle velocity, particle size etc) to a user defined local region within the boiler recorded by the model. These inlet conditions are then used in a subsequent FURNACE simulation of the flow patterns in this local region. As the physical size of the local region is small, a much finer grid resolution (with improved accuracy) can be used for this simulation. Information about particle impacts with individual tubes (particle velocity, particle concentration and impact angle) can be predicted and erosion rates estimated using the empirical tube wear correlation. Enhancement of the code is in progress for Conditional Moment Closure (CMC) modelling of CO and NO_x generation and dispersion/reaction specifically for bagasse combustion.

For sugar processing applications early use was made of the FIDAP code for simulation of juice clarification. Further clarifier development is continuing with the CFX code. CFX has been used extensively for the other processing applications including evaporator circulation and boiling dynamics, vacuum pan boiling and crystallisation which includes three phase flow dynamics incorporating a non-Newtonian fluid, and stirrer design for vacuum pan circulation. Recently extensive development has been undertaken with CFX for the simulation of bagasse gasification in an advanced gasification technology program.

MODEL APPLICATIONS

In this section a range of CFD modeling applications are presented including combustion, clarification, sugar juice evaporator circulation, pan stirrer design and bagasse gasification.

Combustion and Steam Generation

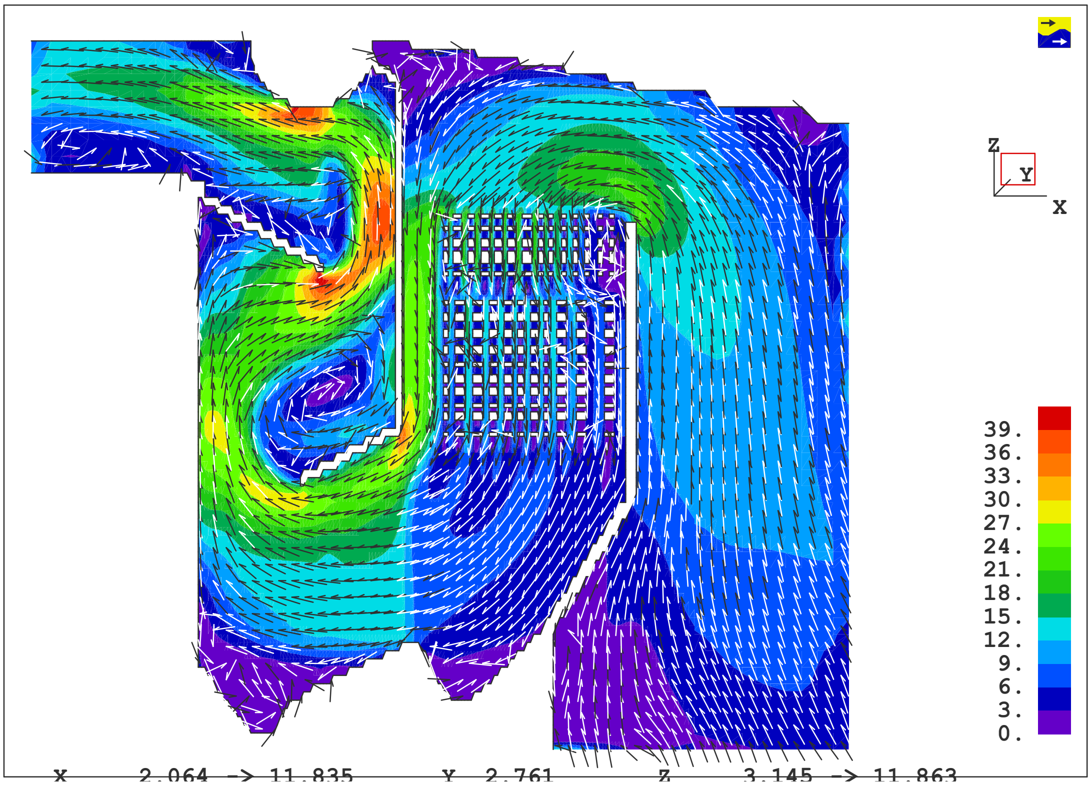
Tube Erosion

Depending on the particular design of the boiler under investigation, the modeling grid is assembled with appropriate porous regions relevant to the problem, such as superheaters, convection banks, airheaters etc. Bagasse fired boilers have convection bank tube configurations that can be susceptible to excessive tube erosion. The tendency of these designs to erode is accentuated by the gas flow dynamics that differentially concentrate the ash particle streams and alter the angle of impact onto tubes into directions that further increase erosion rates. FURNACE simulations of the more intractable problems typically use a dual particle size distribution consisting of representative ‘large’ size/variable density particles for bagasse and ‘small’ dense particles for the ash fraction. Figure 1 shows a typical CFD erosion application in the convection tube bank of a bagasse fired boiler. Both the geometry of the tube rows in the bank and the configuration of flow deflection baffles have been manipulated to arrive at a reasonable resolution of the problem. It has been found that satisfactory improvements in boiler tube erosion performance can be deduced by visual assessment alone of the predicted flow and trajectory patterns. Particular attention is given to particle concentration effects, altered angles of impact and gas velocity distributions. Only in difficult flow conditions is use made of the FURNACE capability to simulate gas/particle flows and calculate erosion rates at specific locations on individual tubes (Novozhilov *et al*, 2001). For several boilers, second stage baffle adjustments have been undertaken to refine flow patterns based on observed changes in erosion patterns and achieve the required erosion reduction. A summary of several tube erosion investigations can be found in Plaza *et al*, 1999.

Convection Bank Heat Transfer

A difficulty that has emerged with tube erosion simulations is the conflict between reduced erosion and reduced convective heat transfer. Heat transfer will be decreased (less degree of crossflow) to achieve improved flow conditions that reduce tube erosion. It has been necessary to include in FURNACE the simulation of global convective heat transfer across tube bundles. This approach was taken to reduce computational effort rather than to model each individual tube in the bundle. Standard correlations for heat transfer to tube bundles have been incorporated [Zhukauskas and Ulinskas, 1988]. These are applied in a framework that utilizes the predictive capability of FURNACE to specify the point values of gas velocity and orientation throughout the porous region that defines the tube bundle.

(a) As-constructed design



(b) Modified design

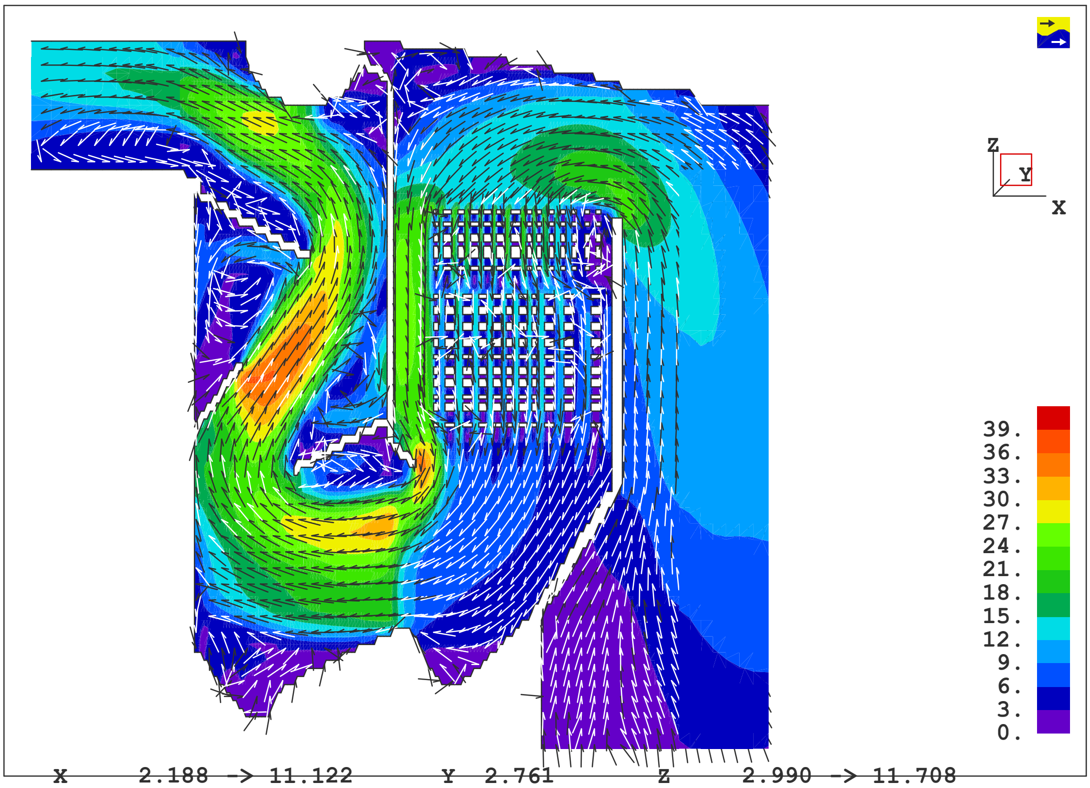


Figure 1: Tube erosion flow simulations for a bagasse fired boiler – gas velocity (m/s).

The global convective heat transfer in the tube bundle is matched to measured data by the use of a scaling factor that is applied throughout the whole porous region. Table 1 shows the results of heat transfer simulations for several bagasse fired boilers. In all cases the convection tube bundles and/or flow baffles have been manipulated to reduce tube erosion. The tabulated gas temperatures are representative values that are the 'mean' values at the exit of the tube bundle but are in reality point measurements. The scaling factor for each boiler has been selected to provide an 'exact' match between the measured and predicted gas temperatures for the original convection bank arrangement. The same factor is then applied for simulations of the modified tube bundles.

Table 1: Measured and predicted convection bank gas exit temperatures (°C) for several boilers before and after convection bank modifications.

Convection bank arrangement	Measured Temp (°C)	Predicted Temp (°C)	Scaling factor
Saint Aubin original	362	363	0.98
Saint Aubin modified	398	396	
Kalamia #1 original	440	440	1.32
Kalamia #1 modified	532	516	
Fairymead #7 original	461	461	1.18
Fairymead 7 modified	454	454	
Kalamia #5 original	271	272	1.35
Kalamia #5 modified	313	312	

Two points are significant. Ideally the scaling factors should be close to 1.0. That factors have been found to depart (in some cases) significantly from 1 is an indication that (i) the global simulation technique for heat transfer across tube bundles is approximate, and (ii) the neglected radiation heat transfer can be masking predictive deficiencies for the convective heat transfer. Note that included within the 'convection' match is the radiant heat transfer that also occurs within the tube bundle. Radiation is not currently modeled specifically within the tube arrays. However it has been observed that for boilers where the scaling factor departs more from unity, these units have more 'abnormal' flow patterns through the tube banks (less clearly defined cross flow segments). Secondly, the use of the global scaling factor does produce very accurate predictions of the overall heat transfer variations in a tube bank after internal flow modifications have been implemented. Predicted temperature differences of less than 5°C are considered to be an excellent result. The simulations show that in most cases the thermal performance of the convection bank has been downgraded (increased gas exit temperature after modification). However downstream heat extraction plant (airheater and economiser) mostly recovers the 'lost' energy such that the overall thermal efficiency of the boiler is not altered significantly.

Airheater Corrosion

Airheater tube corrosion has existed as a major maintenance cost for many years on most boilers. Research [Dixon et. al. 2000] has demonstrated that the problem has its origin in gas and air flow distribution patterns and occurs while a boiler is operating. Measurements of airheater tube temperatures have shown that wall temperatures close to 60°C, less than the dew point temperature, can occur in airheaters where the mean gas exit temperature was greater than 260°C. Negligible corrosion occurs during the start-up phase, which has been the traditional explanation for the problem. It has been determined that the uniformity of the gas flow distribution at the airheater inlet is critical for preventing dew point condensation inside some airheater tubes. This is coupled with the matching air flow distribution, which can act to accentuate the gas flow deficiencies. Global modeling of airheater performance has shown that the overall heat transfer of a tubular airheater can be improved (exit gas temperature reduced) by about 15–20°C by appropriate correction of the gas and air flow non-uniformities.

Secondary Air Injection for Furnace Flame Manipulation

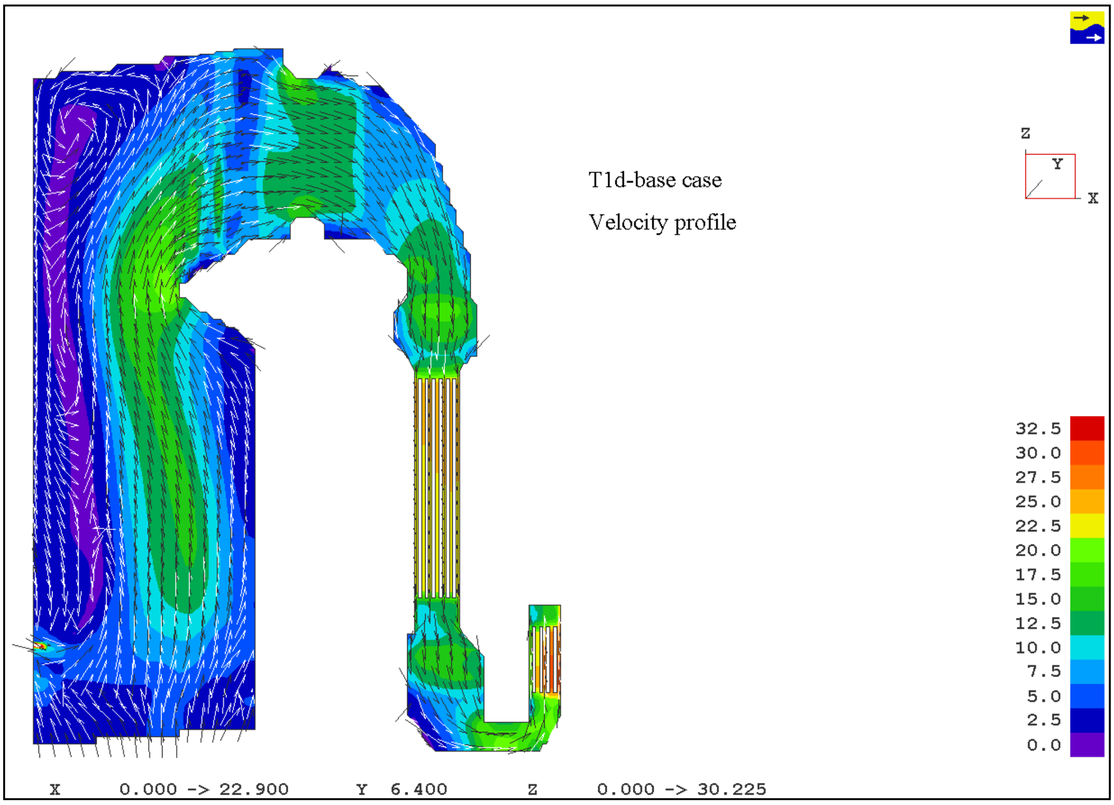
All biomass fired furnaces have secondary air or overfire air injection primarily for CO removal. The secondary air jet curtains also serve to interact with the main flame column generated by the bagasse injection spreaders to maneuver the flame pattern, increase fuel-air mixing and reduce unburnt fuel particle carryover from the furnace. In combination with the swirl spreader development (refer later section) a novel secondary air injection configuration is being investigated as a means of substantially increasing the bagasse firing density within the furnace space. Figure 2 shows one of the several variants of the upper furnace secondary air injection that has been investigated to date. Both simulations have swirl spreader firing.

The conventional secondary air and spreader flow pattern generates a predominant upflow column positioned towards the rear-centre of the furnace. There may be a low strength recirculation flow adjacent to the furnace front wall. The trajectory of the fuel particles through the furnace, which ultimately determines the residence time for burnout, is generally vertically upward with a small flow deflection around the nose at the furnace exit ahead of the superheater section. With the advanced secondary air pattern (b), the flow recirculation within the furnace is deliberately enhanced. This is intended to achieve three objectives; (i) stabilize and anchor the main flow column in the rear 25% of the furnace adjacent to the rear wall, (ii) accelerate the gas flow around the tip of the furnace exit nose, centrifuging the larger partially burnt fuel particles into the recirculation flow, and (iii) establish the large, high strength recirculation flow adjacent to the front wall. This flow serves a dual purpose of returning the separated unburnt particles to the main flame zone and supplying higher temperature gases to the entrainment flow of the bagasse spreaders to increase the particle drying rate and improve ignition stability.

Secondary Air Distribution for CO Minimisation

Carbon monoxide (and NO_x) emissions are emerging as pollutants of concern for boiler design and operations in the Australian sugar industry. Recent investigations with FURNACE have been directed to quantifying the generation and removal characteristics of carbon monoxide and how these can be manipulated by the use of secondary air or overfire air injection. The code modifications associated with this work are a companion to the CMC development mentioned earlier. The CO modeling has been focused on the mean residual levels of CO that are ejected from a furnace and the distribution of CO in the upper furnace region in the area where urea injection for NO_x control is implemented.

(a) Conventional furnace configuration



(b) New furnace configuration.

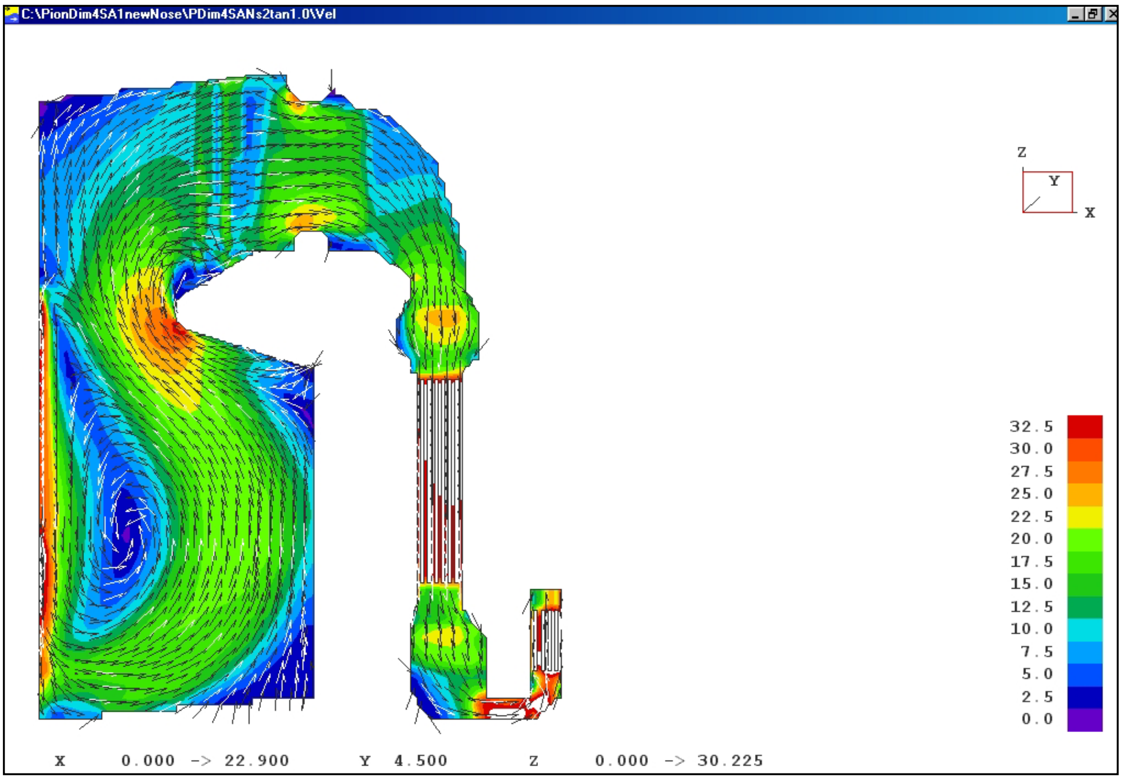


Figure 2: Advanced secondary air injection for furnace uprating. – gas velocity (m/s)

Incorporation of the CO dynamics in the FURNACE code has been implemented by assigning a fraction of the off-gas produced during both devolatilization and char burnout to be CO. The 'CO fraction' factor has been selected by comparing the predicted value of the mean CO concentration at the boiler convection bank exit with operating plant measurements. The reaction between CO and O_2 to form CO_2 was approximated by a modified version of the eddy break up model. This takes into account the kinetic limitations of the CO oxidation reaction at low temperatures and the mixing limitations of the reaction at higher temperatures. The modified eddy breakup model helps represent the behaviour of 'parcels' of CO that are much smaller than the computational cell size used in three dimensional models of full scale boilers.

Investigations have focused on examination of the number, location, size, orientation, injection velocities and mass flows of the overfire system on a boiler design. The assessment of the success of a particular overfire air configuration is based on the visual evaluation of the flow and mixing patterns of the overfire air jets with the bulk furnace gases, the manipulation of the main flame column up through the furnace, the variations in the CO contours within the furnace and the mean concentration of CO present in the boiler discharge flue gas. Typical predictions show high levels of CO in the devolatilisation and ignition region in the lower section of the furnace (>12,000ppm) consistent with in-furnace flame zone measurements. The CO is oxidized rapidly within the main flame zone and a residual column of low concentration CO flows up the centre of the furnace where the flame column is positioned. It is the manipulation of this residual CO stream in the upper furnace that is the focus of the overfire air jet variations. A recent study predicted the mean CO concentration at the superheater zone exit to be 131 ppm with the optimized overfire air configuration and 427 ppm when firing with zero overfire air injection.

Ignition Stability and Swirl Burner Technology

Recent designs of boilers installed in several Australian sugar mills have experienced a combustion instability that is manifest as a periodic oscillation of the combustion rate of bagasse deposited on the grate at the bottom of the furnace. The cause of the instability appears to be excessive energy extraction from the bagasse ignition region above the grate resulting from water-cooling of the grate structure. The instability is observed as oscillations of the quantity of bagasse deposited combined with matching variations of flame luminosity, furnace pressure, flue gas oxygen and carbon monoxide concentrations and steam pressure. The magnitude of the instability can be such that physical damage to the boiler structure occurs (failure of airheater walls, air duct separation, expansion joint fracture) and boiler safety is compromised. The FURNACE code has been modified by the modeling of the bagasse deposition and burning process on the grate, including particle segregation, deposition on the grate surface, moisture removal from the accumulated pile and surface combustion. The code is able to simulate the oscillation dynamics with sufficient accuracy to explore the boiler design and operating factors that impact the process [Woodfield *et al*, 1999].

An area of continuing CFD enhancement is the application to the ignition stability issues of the high intensity swirl burner technology developed by SRI [Dixon and Martel, 1997]. CFD was not used during the development sequence for the swirl burner technology but is needed to refine the burner design. The burner exhibits interesting stability behaviour that requires further investigation and is well suited to CFD exploration. The technology allows the variation of the moisture content of the fuel such that the burner flame can become unstable and exhibit partial detachment. Under these conditions the combustion intensity within and surrounding the central recirculation zone progressively decreases, flame luminosity reduces and the near flame field becomes visibly transparent with weak flame pockets attempting to remain alight. There is no sudden extinction of the flame. Due to the proximity of adjacent flames, if the swirl burner flame does become fully extinguished and the bagasse moisture content is then reduced, reignition occurs via a reversal of the process just described. It is necessary that a more complete understanding of the ignition dynamics of the swirl burner is gained so that appropriate design and operating parameters can be determined. It is considered that the conditions of particle drying and ignition in the near burner field will provide an interesting challenge when the CFD modeling is undertaken.

Advanced Swirl Spreader Technology

Development is progressing for an advanced swirl spreader device for enhanced bagasse combustion. The swirl spreader is designed to replace the current technology linear pneumatic spreader. The enhanced mixing and lateral spreading of the fuel generated by the swirl action provides opportunities for boiler design refinements that could result in significant capital cost reductions for boiler plant. Two of the significant boiler design factors associated with the swirl spreaders (compared to conventional technology) are a 50% reduction in the number of spreader units required for a given firing capacity, and increased drying and burning rates leading to increased bagasse firing rates within a given furnace volume. The swirl spreader technology has been developed to the stage of large scale application on an operating boiler.

CFD has been used to optimize the internal aerodynamics of the spreader design and to resolve erosion problems due to unwanted particle recirculation in a critical area. A process of simulation matching of the observed flame characteristics of the swirl spreaders has been completed for measurements on the test boiler at Proserpine mill. Exploration of the design consequences of swirl spreader integration in a furnace is in progress. CFD is a critical element in the full development sequence for the swirl spreader technology. It is being applied to quantify the full operating envelope of the spreaders including the interactive effects that derive from spreader integration with the furnace. The modeling also forms an integral stage of the engineering design of the advanced boiler configuration of which the swirl spreaders are a significant component.

Gasification

SRI is currently involved in a joint project with Hokkaido and Monash Universities to develop a novel gasifier that utilizes the catalytic effects of alkali species inherent in biomass feedstocks to produce a high quality (low tar) product gas at relatively low temperatures (500^oC to 700 ^oC). The task of SRI relates primarily to the development of appropriate numerical simulation models with which fundamental experimental data can be extrapolated to determine the performance and viability of a thermodynamically optimised, prototype-scale, gasification plant. To this end a detailed CFD representation of the hydrodynamics, thermal energy transfer and chemical reactions associated with processes taking place in the fluidised bed gasification reactor has been developed. The CFD model accepts data such as species concentration and temperature-dependant chemical reaction rates, bed and fuel material properties, gas properties and vessel geometry. As such it represents a tool that when validated will be used at the design stage of the prototype-scale gasifier reactor.

A vertical, tubular gasifier reactor has been simulated. In this model, steam and bagasse are introduced at the base of the lower entry section. Mixing of and mass transfer between the bed material and reactants takes place in a bubbling fluidised bed in the lower region of the reactor. Gaseous and solid (char) products together with small amounts of the inert bed material are carried out through the top surface of the exit section. Energy is delivered to the reactants in the fluidised bed from an external source via internal heating surfaces. Currently this stage is represented simplistically as isothermal reactor wall conditions.

A CFD model of the biomass reformer has been developed on the newly released CFX version 5.6. A fully Eulerian approach to simulating all the phases within the gasifier has been adopted. The model can be described as both multi-phase and multi-component. The CFX code is configured to recognize different components as belonging to a single phase when those components have a common velocity field. This treatment (using common velocity fields) provides the opportunity for considerable computational savings relative to an individual velocity calculation for each component. A summary of the phases and components used in the current model are given in table1. As either evaporation of moisture or chemical reaction progresses in the reactor, mass is transferred between phases or between components within the same phase or both.

Table2: Phases and components in the current simulation

Phase	Component(s)	Eulerian fluid type
Bed material	Silica	Granular, dispersed
Biomass	DAF Bagasse, ash, water	Granular, dispersed
Char	Carbon	Granular, dispersed
Gases[1]	H_2O (steam), CO_2, H_2O, CH_4, H_2	Conventional, modified ideal gases

Some of the phases used in the model are solids. These phases are represented as modified or 'granular' fluids dispersed through the more conventional fluids (gases) in the reformer vessel. The (Eulerian) granular fluid representation of the solid phases permits characteristics of the dispersed solid (particle size, shape, density etc) to be implemented in the fluidised material. These particle characteristics influence bulk properties of the granular fluid including density, viscosity and turbulence. This has distinct advantages over the Lagrangian approach of representing the solid phases as a large number of individual particles and is used increasingly in the representation of fluidised beds (see for example Boemer [1997], Guenther et al [2001], Ibsen et al [2001]). In particular the alternative Lagrangian models do not account for interaction between particles, a feature that would introduce significant inaccuracies at the dense phase conditions existing in the fluidised bed region of the reformer. The current CFX code applies an additional pressure term (see Gidaspow, in the solid phase NS momentum equation) to account for collisions between solids under dense phase conditions.

The flow regime within the main body of the reactor vessel is turbulent for the range of biomass fluidising gas velocities (0.9 to 4 $m.s^{-1}$) considered typical [Anon, 1994] of bubbling beds. In dispersed two-phase flows, large particles tend to increase turbulence in the continuous phase due to the presence of wakes behind the particles. The current model takes account of this by utilizing a modified k-e approach developed by Sato and Sekoguchi [1975] to particle induced turbulence.

The nature and sequence of reactions relevant to the gasification of biomass are well documented. In this respect, the analysis used in the current model draws heavily on Fletcher et al [2000]. The four basic processes simulated are those of initial drying and devolatilization and subsequent heterogeneous and homogenous reactions. It is assumed that the process of devolatilization is energetically neutral [Di Blasi, 1993] ie. there is equivalence in terms of the energy value of the original biomass and the products of devolatilization. This assumption together with a mass balance on the reactants and products of the devolatilization process enables a unique composition to be determined for the vapour products. A spreadsheet model has been developed in which the heating value of bagasse is compared with a series of combustion calculations on the products. An optimising solver varies the mass fraction of each of these products until mass and energy equivalence between the original bagasse and final products is achieved. The resulting gas composition is input into the CFD model as the products of the pyrolysis process. In terms of the rate at which volatiles are produced this is taken from Fletcher et al [2000]. This rate is proportional to the instantaneous mass of remaining volatiles, time and temperature.

The heterogeneous gasification reactions currently implemented are: $C + CO_2 \neq 2CO$, $C + H_2O \neq H_2 + CO$ and $C + 2H_2 \neq CH_4$. The associated reaction rates are determined as the minimum of the kinetic and diffusion rates where the rate constants are taken from Wen and Chaung [1979] based on coal gasification. Simple global reactions are used to describe the homogonous gas phase chemistry. Reaction rates are taken from data published by Jones and Lindstedt [1988]. The gas phase reactions currently implemented are: $H_2O + CO \equiv H_2 + CO_2$ and $CO + 3H_2 \equiv CH_4 + H_2O$

The hydrodynamics of a fluidised bed are inherently unstable. For this reason the current model is cast in a

[1] The component set currently included is not considered exhaustive. Other components such as tars will be added as a greater understanding of their respective chemistry is developed.

transient form. A sample output from the gasification model at the onset is shown in figure 3.

Figure 3: Predicted mass fraction of solid fuel volatiles in the gasifier vessel.

Current activities are focused on the implementation of experimental reaction yield and kinetics data for biomass reforming under catalytic conditions. The results from this CFD model will provide inputs to a process model. The latter has been developed to simulate material and energy flows between individual components of a practical operating gasification cycle. The primary function of the process model is to quantify the necessary transfer of energy and material between components such that sustainable and ultimately optimal operation of the gasification cycle is achieved.

Sugar Processing

Vacuum pan stirrer design

A numerical model for the liquid flow inside a sugar mill batch vacuum pan vessel including mechanical agitation only has been developed. The model assumes the major driving force for circulation within the pan is due to the agitator and (for now) assumes the natural convection flow effects are negligible. A series of physical scale model experiments were performed by CSIRO to obtain data on the flow field using LDV techniques. The data then formed the basis for comparison with CFD model predictions. The CFD model showed very good agreement with the measured velocities from the scaled model. The modelling tools were then applied to agitator blade geometries to identify an optimised design.

The major design criteria were focussed on maximising the volumetric flow rate within the vessel, with lower weighting assigned to reducing shaft power requirements and eliminating localised eddies in the fluid flow. Figure 4 shows a typical plot of the flow rates generated by the different stirrer blade designs used in the evaluation. An innovative agitator blade design has been developed and modelled extensively on a full scale test vessel. The new design agitator will be installed in a test vessel in early 2004 and a series of velocity measurements will be taken to compare with the model predictions and confirm the design.

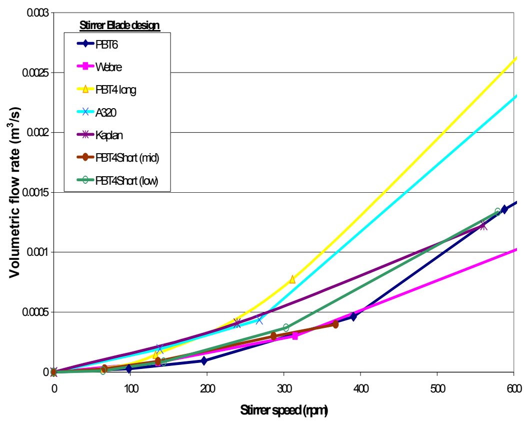


Figure 4: Comparison of flow rate generated by stirrer blade design.

Evaporator circulation and boiling

A numerical model for the single-phase fluid flow inside a sugar mill evaporator vessel has been developed. The model uses experimentally measured correlations to describe the changing sugar juice properties as the temperature and concentration changes throughout the flow field. The present model is limited to only the liquid phase but allowances for the production of vapour due to boiling have been made. The entire geometry of the vessel has been modelled including the vertical heating tubes in the calandria section. The calandria is modelled as an open space but the effect of the tubes is simulated mathematically by the inclusion of a momentum source in three directions. Vertical flow resistance is modelled as the pressure drop of a single phase laminar flow in a pipe.

As the water component of the mixture is converted to vapour the remaining sugar solution is progressively concentrated. The increase in sugar concentration is calculated at each node by performing a mass balance around the evaporation process. The production of vapour is modelled as a mass sink for the water component within the liquid. The evaporation process is linked to the heat flows by means of switching off the mass sink until the fluid temperature reaches the appropriate saturation temperature. However, once evaporation starts it is assumed to be constant and equal to the average evaporation rate of the entire vessel. No allowance has been made for the non-uniform energy flows (and evaporation rate) between calandria tubes or the relationship between evaporation rate and sugar concentration. The energy flow into the fluid is defined as the amount of sensible heat required to increase the liquid temperature due to boiling point elevation. The energy that enters the fluid domain and drives the evaporation process is neglected since this amount of latent heat has no influence on the liquid phase. This assumption is correct provided the predicted evaporation rate is correct. The current assumptions about the distribution of energy are necessary in the absence of any further knowledge about the fluid flows on the steam side of the calandria tubes. Figure 5 shows a typical prediction of the velocity field within the body of the evaporator. The calandria body is indicated.

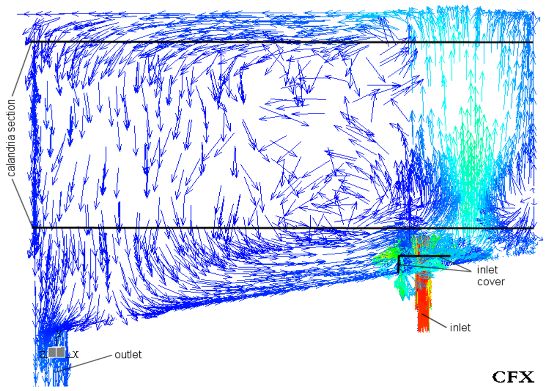


Figure 5: Vector plot of the flow field inside an Evaporator vessel.

Juice clarification

SRI developed a high capacity juice clarifier in 1969 that has been used extensively in the world sugar industry to deliver improved juice and sugar quality with reduced lime and flocculant consumption. During the early 1990's as SRI expanded its activities with CFD applications in sugar manufacturing, FIDAP was used to undertake a preliminary evaluation of the fluid dynamics of the standard SRI clarifier. This analysis produced surprising results that showed multiple high velocity regions causing unwanted agitation and hindered mud settling rates, poor use of the available flow area and opportunities for design and performance improvements. Development of the new generation (NG) clarifier was initiated in 1996. CFD was used extensively to explore modified design concepts that led to the latest design. The use of a novel perforated plate across the top of the clarifier vessel has allowed capacity increases of over 75% compared to the standard design. Other design improvements have evolved with the juice feed distribution system, the deflector design in the feed launders and the mud scraper system to minimize settled mud reintrainment.

Fifteen new clarifier designs have been installed in Australian and overseas sugar mills, with reported benefits of increased clarification quality, throughput and sugar quality with reduced turbidity, bagacillo carryover and flocculant usage. Figure 6 shows a FIDAP velocity plot of the new clarifier configuration.

CONCLUSIONS

Major advances have been made in the understanding and development of bagasse combustion technologies and other sugar manufacturing processes . CFD modeling has come to form an integral and critical part of this progression. The experience with CFD in the sugar industry through SRI has encompassed the full range of applications from fundamental code development, through the generation and commercialization of new ideas and technologies, to the resolution of practical plant problems. The paper has summarized several of these applications where SRI has achieved successful results. It has been illustrative for SRI that the full benefits of CFD in the delivery of commercial outcomes, be it in new technologies or the solution of operating plant problems, have emerged through the close interaction between the code development and the validation via full scale plant simulation. It is fundamental to the applicability of CFD that regular and routine validation with full scale plant data is undertaken. The two-way interaction between development and application enhances the code fundamentals by focusing on practical issues and the need to resolve these effectively. In some cases the physical size of the problem areas that are being investigated, within a commercial plant environment, can be very small. Similarly the confidence in the capabilities and 'truth' of the CFD predictions at the commercial scale is enhanced, to the extent that firm engineering decisions are made based solely on the characteristics predicted by the code.

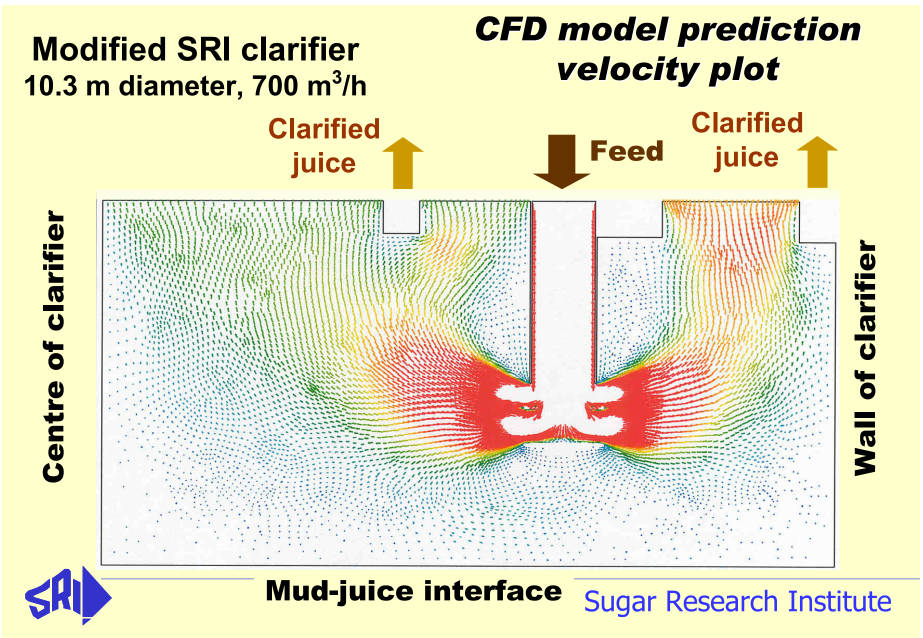


Figure 6: Liquid velocity predictions (m/s) for a typical NG clarifier design.

SRI now regularly applies CFD modeling to a wide variety of flow and combustion related areas as a routine investigation and design tool. This notwithstanding, it has been surprising to SRI that there remains a continuing suspicion of the capabilities and value of CFD modeling to plant design and process understanding in the sugar industry by both plant operators and equipment vendors.

ACKNOWLEDGEMENTS

Funding for the advanced swirl spreader and secondary air technologies was provided in part by the Australian Greenhouse Office (through RECP) and the Sugar Research and Development Corporation. The Japanese Government through the NEDO program is acknowledged for the funding of the gasification project. The Queensland Department of State Development is acknowledged for their contribution to the funding of the evaporator and vacuum pan development projects.

REFERENCES

ANON, (1994), "Fluidised bed combustion and gasification; a guide for biomass waste generators", US Department of Energy publication available from http://www.ott.doe.gov/biofuels/pedownload.html.

BAKER, E.G. BROWN, M.D. and ROBERTUS R.J. (1983), "Catalytic gasification of bagasse for the production of methanol", Final report prepared for the Sugar Research Institute under contract 2311257000, Battelle Pacific Northwest Laboratories, Richland, Washington 99352, (report may be available from SRI on request).

BOEMER, A., QI, H. and RENZ, U. (1997), "Eulerian simulation of bubble formation at a jet in a two-dimensional fluidised bed", Int J. Numerical Methods Eng., 23, 927–944.

BOYD, R.K and KENT, J.H., (1986), "Three-dimensional furnace computer modeling", 21st Symposium (International) on Combustion. The Combustion Institute, 21, 265–274.

CHEN, W.Y. and CHAUNG, T.Z., (1979), "Entrained coal gasification modelling", Ind. Eng. Chem. Process. Dev., 18, 4, 684-690.

DI BLASI, C., (1993), "Modelling and simulation of combustion processes of charring and non-charring solid fuels", Prog. Energy Combust. Sci., 19, 71-104.

DIXON, T.F and MARTEL, H., (1997), "Commercial Applications of the SRI Swirl Burner Combustion System", Proc. Aust. Soc. Sugar Cane Technol., 19, 456-465.

DIXON, T.F., PLAZA, F and MANN, A.P., (2000), "Prevention of Airheater Corrosion", Proc. Aust. Soc. Sugar Cane Technol., 22, 491-497.

FLETCHER, D.F., HAYNES, B.S. CHRISTO, F.C. and JOSEPH, S.D. (2000), "A CFD based combustion model of an entrained flow biomass gasifier", Applied Mathematical Modelling, 24, 165-182.

GUENTHER, C. O'BRIEN, T. and SYAMLAL, M., (2001), "A numerical model of silane pyrolysis in a gas-solids fluidised bed", Proceedings of the 4th International Conference on Multiphase Flow, New Orleans, LA, E. Michaelides Eds.

IBSEN, C.H. SOLBERG, T. and HJETAGER, H., (2001), "Evaluation of a three-dimensional numerical model of a scaled circulating fluidised bed", http://hugin.auc.dk.

JONES, W.P. and LINDSTEDT, R.P., (1988). "Global reaction schemes for hydrocarbon combustion", Comb. Flame 73, 233-249.

LUO, M.C., (1993), "Combustion of bagasse in a sugar mill boiler", Ph.D. thesis. University of Queensland, Australia.

MANN, A.P., PENNISI, S.N., DIXON, T.F., NOVOZHILOV, V., KIRKPATRICK, M., KENT, J.H., SAZONOV, V. and ZHANG, L., (2001), "Modeling of boiler tube erosion", Proc. Aust. Soc. Sugar Cane Technol., 23, 369-375.

MANN, AP., PLAZA, F. and DIXON, T.F., (2003), "Improving the prediction of convection bank heat transfer", Proc. Aust. Soc. Sugar Cane Technol., 25.

NOVOZHILOV, V., KIRKPATRICK. M., KENT, J.H., SAZONOV, V., ZHANG, L., MANN, A. and DIXON, T., (2001), "Computational fluid dynamics modeling of tube erosion rates in bagasse fired boilers", Journal of Institute of Energy, 74, 57, June.

PLAZA, F., DIXON, T.F., DICKENSON, N.L., FITZMAURICE, A.L. and OWENS, M., (1999), "Performance of baffled boilers with redesigned convection banks", Proc. Aust. Soc. Sugar Cane Technol., 21, 432-437.

SATO, Y. and SEKOGUCHI, K., (1975), "Liquid velocity distribution in two phase bubbly flow", Int. J. Multiphase Flow, 2, 79.

WEN, C.Y. and CHAUNG, T.Z. (1979), "Entrainment coal gasification modelling", Ind. Eng. Chem. Process Des. Dev., 18:4.

WOODFIELD, P.L., KENT, J.H. and DIXON, T.F., (1998), "Computational modeling of a bagasse-fired furnace – effects of fuel moisture", Proc. Aust. Soc. Sugar Cane Technol., 20, 458-464.

WOODFIELD, P.L, KENT, J.H. and DIXON, T.F., (1999), "Computational Modeling of Combustion Instability in Bagasse Fired Furnaces", Mediterranean Combustion Symposium, Antalya, Turkey, 813, June.

WOODFIELD, P.L. (2001), "Combustion instability in bagasse-fired furnaces", Ph.D. thesis, University of Sydney, Australia.

ZHUKAUSKAS, A.A. and ULINSKAS, R., (1988), "Heat Transfer in Tube Banks in Cross-flow", Springer, New York.