Fifth International Conference on CFD in the Process Industries
CSIRO, Melbourne, Australia
13-15 December 2006

CFD MODELLING OF DIRECT CONTACT STEAM INJECTION

Curtis MARSH[1] and Denis WITHERS[2]

[1]CFD Design & Engineering Ltd, PO Box 30003, Christchurch 8246, New Zealand
cmarsh@engineeringCFD.com
[2]Aughinish Alumina Ltd, Askeaton, Ireland

ABSTRACT

The direct injection of steam into a process stream is a method of heating used in many process industries. The amount of research in this area however is limited to the nuclear industry, with applications relating to reactor cooling systems. There are no general CFD models available for designs relating to the process industry.

A lab-scale experiment was conducted to investigate the direct injection of steam into water. The height of the steam plume was measured against different water temperatures providing a means to validate the subroutine used in the CFD models.

A subroutine was developed to describe the heat and mass transfer between the vapour and liquid phases, based on previous work by the author on flashing flows. The subroutine was compiled into FLUENT (ver 6.2.16) using the Eulerian multiphase model.

The model was then applied to an industrial scale problem giving unique insights into the operation of the equipment and the behaviour of the condensing steam.

Key words: Steam; two phase flow; condensing; injection; heat and mass transfer

NOMENCLATURE

A	1/m	interfacial area
a	-	volume fraction
C_p	J/kg-K	specific heat
h	W/m^2-K	heat transfer coefficient
i_{lat}	J/kg	latent heat
Ja	-	Jacob number
J	1/m^3-s	source term
k	W/m-K	thermal conductivity
N	1/m^3	bubble number density
Nu	-	Nusselt number
r	kg/m^3	density
R	m	radius
T	K	temperature
Subscripts		
b		bubble
E		energy
l		liquid
M		mass
sat		saturation
v		vapour

INTRODUCTION

The injection of steam into a liquid is a direct heating method used in many process operations and has several advantages over other means of heat transfer, such as not being affected by the degree of superheat, heat transfer does not deteriorate with scaling and large increases in temperature can be achieved over very short time scales.

The overall energy balance for this process is readily calculated with the properties of steam well defined over a large temperature range. The behaviour of steam as it condenses while in direct contact with a sub cooled liquid is not so well defined, with research limited to the nuclear industry.

Lee and No (1998a) present results and theories relating to nuclear reactor cooling systems using the RELAP code, developed for calculating thermal- hydraulic transients in water-cooled nuclear reactors. Chun Kim and Park (1996) present experimental results of steam injected into subcooled water and characterise plume shapes for different steam mass flux. Lee and No (1998b) also present experiment data of steam experiments and have published condensation regime maps describing three regimes, steam cavity, chugging and sub-sonic jetting.

While commercial CFD codes can readily model liquid vapour systems there are currently no subroutines or published theories that allow this particular heat and mass transfer process to be accurately modelled.

The performance of a high pressure steam injection system at Aughinish Alumina Ltd was giving variable results with very small changes in process conditions. A project involving both experimental work and numerical simulations was undertaken to better understand the dynamics of the direct contact steam condensing process, with the aim of creating a practical approach of modelling this process with CFD and designing an improved steam injection system. The key aspect identified for this work was that the mixing of the fluid is principally buoyancy driven and the rate of steam collapse is a key variable.

MODEL DESCRIPTION

Condensing Steam Model

The interface between the liquid and vapour is rapidly changing and contains both large and small surface features and bubbles, over which the heat and mass transfer takes place. It is not practical to model the small flow features with CFD as the required grid size and the time steps would be prohibitively small and not practical for industrial applications.

To calculate the rate of heat and mass transfer the theory from previous work modelling flashing flows (Marsh, 2004) was used as a starting point. The rate of energy transfer is based on three key parameters; interfacial area, heat transfer coefficient and the driving force ?T.

Interfacial Area

The interfacial area is the surface area between the vapour phase and the liquid phase over which the energy and mass transfer takes place. This is in effect the total surface area of the vapour bubbles and is expressed in terms of area per unit volume. The approach used by Blinkov et al (1993) determined the number of bubbles and then calculates the radius of the bubble based on the local vapour fraction. For spherical bubbles, the interfacial area ($A_{i,b}$), and the local vapour fraction (α) are shown in equations 1 and 2 respectively.

$$A_{i,b} = 4\pi R_b^2 N_B \quad (1)$$

$$\alpha = \tfrac{4}{3}\pi R_b^3 N_B \quad (2)$$

Combining these equations gives an equation (3) for the interfacial area.

$$A_{i,b} = 3\alpha / R_b \quad (3)$$

When considering flashing flows the bubble number density is determined from bubble nucleation theory. However with the process of collapsing steam the bubbles are formed by the break-up of larger bubbles and slugs. As there is no information available for this variable for condensing flows, a constant bubble density per unit volume was assumed and the value based on the experimental work that follows.

Heat Transfer Coefficient

The heat transfer coefficient between the liquid and vapour phases was calculated using the following equations;

$$Ja = \frac{c_{p,l}\rho_l \Delta T}{\rho_v i_{lat}} \quad (4)$$

$$Nu = \frac{12Ja}{\pi}\left[1 + \tfrac{1}{2}\left(\frac{\pi}{6Ja}\right)^{2/3} + \frac{\pi}{6Ja}\right] \quad (5)$$

$$h = \frac{k_l Nu}{2R_b} \quad (6)$$

The formulation for the heat transfer coefficient h is based on the Nusselt number Nu and the Jacob number Ja.

Driving Force

The driving force for the condensation of steam is the difference between the local liquid temperature and the liquid saturation temperature.

$$\Delta T = T_{sat,l} - T_l \quad (7)$$

Energy and Mass Source Terms

The energy transfer was assumed to act only across the interfacial area and thus the total energy transfer was calculated from the product of the driving force ?T, interfacial area and the heat transfer coefficient. The total energy transferred was then used as the source term (J_E) in the energy transport equation for each phase within the multiphase model in the following form.

$$J_E = hA_i\Delta T_l \quad (8)$$

As the liquid cannot exceed the local saturation temperature without changing back to steam this source term was only considered to act when the liquor temperature was below the saturation temperature. The phase change from liquid to vapour was not considered to be significant.

As the superheated steam reduces in temperature and reaches the local saturation temperature, the mass transfer process begins. The mass source term (J_M) being equal to the energy source term divided by the latent heat of condensation, where the steam temperature is equal to local saturation temperature.

$$J_M = \frac{hA_i\Delta T_l}{i_{lat}} \quad (9)$$

EXPERIMENT

The theory presented above allows the direct contact condensing process to be defined, however it leaves one variable undefined, the density of bubbles used to calculate the interfacial area between the phases. To determine this parameter and to visualise the actual condensing process the following experiment was devised.

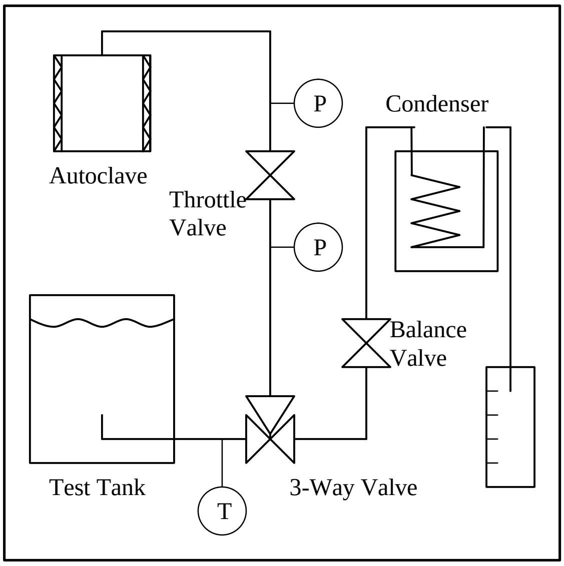

Figure 1: Schematic of Experiment

Figure 2: Experimental Layout.

An autoclave with a 1 gallon capacity was used as the steam source allowing a closely controlled set point of steam temperature to be achieved. The steam flow was controlled by a fine needle valve and high accuracy pressure transducer.

To calculate the mass flow of steam a three-way valve was used to direct the steam to a condensing coil, allowing the volume of condensate to be directly measured over time. A second needle valve (balance valve) was used to ensure that the pressure drop through the condensing coil was equivalent to the test vessel, confirmed by verifying a consistent pressure drop across the first valve.

When steady state conditions were established the steam was introduced to the test vessel through a 6mm stainless steel tube allowing the condensing process to be observed. An example of the steam plume is shown in figure 3.

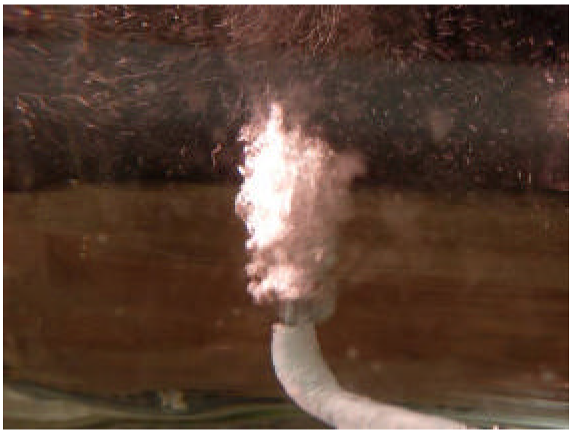

Figure 3: Steam Plume

The temperature of the steam and water were varied to gain an understanding of the different vapour collapsing regimes and to better understand the relationship between the steam plume and the degree of tank mixing.

While holding the steam temperature and flow rate constant and varying the liquid temperature the height of the steam plume could be measured. This relationship is shown in figure 4 and follows an exponential relationship.

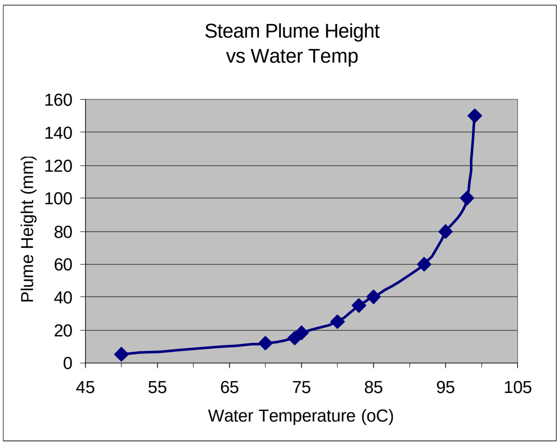

Figure 4: Relationship between plume height and steam temperature.

This relationship was used as a means to determine a representative value for the bubble density allowing the correct interfacial area between the phases to be determined, completing the mathematical model of the condensing process.

IMPLEMENTATION OF THE NUMERICAL MODEL

The presented theory was implemented within Fluent (version 6.2.16) through the use of a user defined subroutine and solved as an unsteady time dependant problem.

There are several multiphase models that could be employed to simulate the two phase system of a sub-cooled liquid and steam, such as the Volume of Fluid (VOF) model, mixture model or the Eulerian model. The VOF model is not suitable as the interface between the two phases is rapidly changing and not always well defined. The Eulerian model was chosen over the mixture model as each phase is treated separately, allowing the energy of each phase to be considered and allows for the proposed subroutine to be easily linked in to the solver parameters.

Validation

A 2-D axi-symmetric simulation was created, equivalent to the experimental test tank, with steam flow at 1 kg/hr and a temperature of 105°C. By implementing the theory presented above and assuming an arbitrary bubble density a transient solution was used and the mean bubble height determined. The bubble density was varied and the simulation repeated to determine the most appropriate empirical value for bubble density. With this empirical value determined, simulations were repeated for a range of temperatures. The resulting bubble height profile can be compared to the experimental data as shown in figure 5.

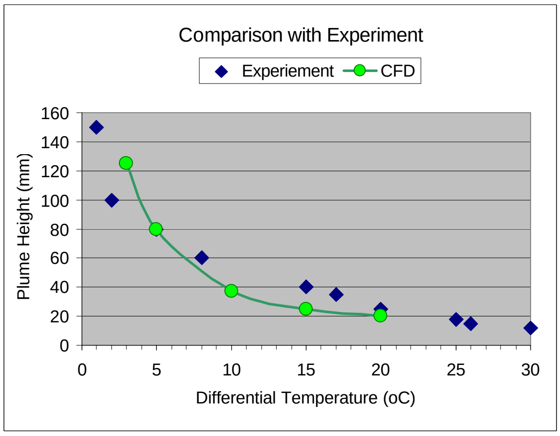

Figure 5: Calculated plume height versus experimental data.

Considering the random variation in the bubble height in both the experimental and numerical results the proposed model was considered suitable for the proposed application. As all the key parameters for the subroutine, with the exception of the bubble density, are calculated from known material properties, it is assumed that the application to a much larger scale problem will have acceptable accuracy.

APPLICATION TO PROCESS PLANT

This model was used to model the injection of steam into a digestion vessel at Aughinish Alumina. The Digester is 4m in diameter and over 30m tall. The steam is injected into the vessel at 310°C at a rate of 350 t/hr at a point below the liquid surface. Colder liquor is added in this region at 200°C and is heated to around 250°C. The steam is superheated by around 50°C and comes into contact with the sub-cooled caustic liquor. The steam quickly looses the superheat and rapidly condenses.

An unstructured grid was developed for the geometry, including the internal nozzles and baffle plates. A surface boundary layer grid was used, completing the mesh with tetrahedral cells in the interior. The total cell count was 650,000 cells.

The solution was strongly transient in nature and required time steps of 0.001 seconds, taking 1 week to solve 10 seconds of solution time.

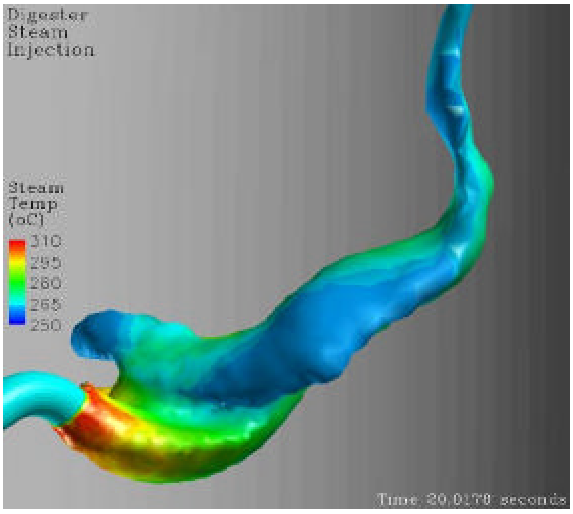

Figure 6: Steam Plume after 20 seconds.

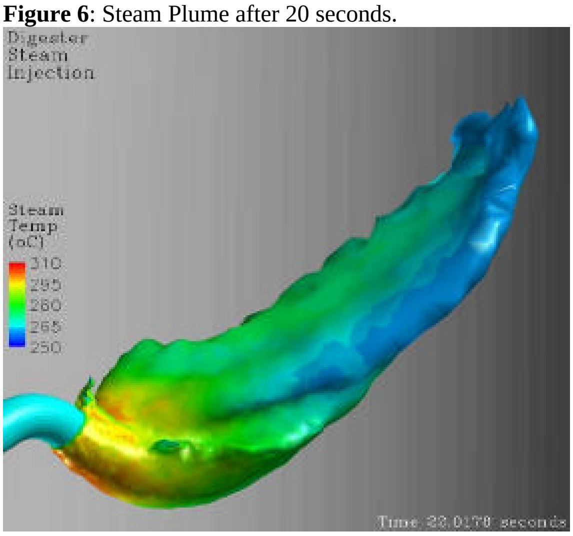

Figure 7: Steam Plume after 22 seconds.

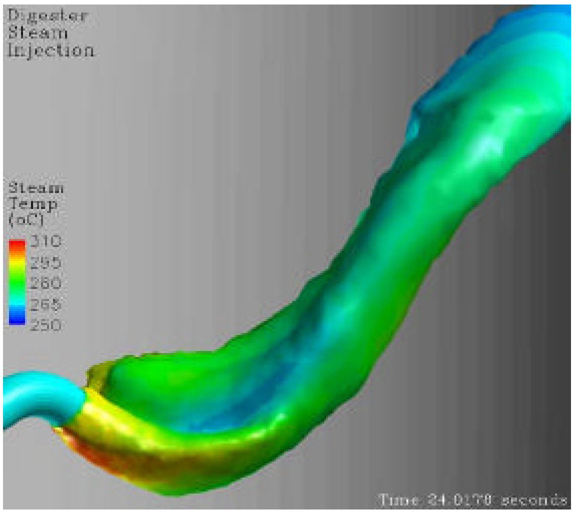

Figure 8: Steam Plume after 24 seconds.

An iso-surface of 50% steam volume fraction is shown in figures 6, 7 and 8, coloured by steam temperature. These results give an idea of how transient the solution is, with the steam plume strongly influenced by the flow of liquor and the relative temperatures of the phases.

Application of the new steam condensing model was successfully applied to an industrial problem and allowed several design concepts to be evaluated. These results also gave new insights into the operation of the equipment and improved understanding of how process factors affect the rate of steam condensation

CONCLUSION

Existing heat and mass transfer theories were reviewed and adapted for the application of condensing steam. The new direct contact steam condensing model was incorporated into a user defined subroutine for inclusion into a commercial CFD software package. The theory required one empirical constant to be defined to complete the mathematical model and this was obtained through experimental work.

The results of the experimental work provided insight into the condensing process, by illustrating the different condensing regimes. The experimental results also allowed the numerical model to be validated, with the results in reasonable agreement over the temperature range considered.

Application of the new steam condensing model was successfully applied to an industrial problem and allowed several design concepts to be evaluated. These results also gave new insights into the operation of the equipment and improved understanding of how process factors affect the rate of steam condensation.

ACKNOWLEDGEMENTS

The author would like to thank the Aughinish Alumina for permission to publish this work and their laboratory staff for their kind assistance with the experimental work.

REFERENCES

BLINKOV, V.N., JONES, O.C., NIGMATULIN, B.I., (1993), “Nucleation and flashing in nozzles-2. Comparison with experiments using five-equation model for vapour void development.” *Journal of Multiphase Flow* 19(6): 965-986.

CHUN M.H., KIM Y.S., PARK J.W., (1996) “An investigation of direct condensation of steam jet in subcooled water” *Int Comm Heat Mass Transfer* 23(7) 947-958.

LEE, S.I., NO H.C., (1998b) “Improvement of direct contact condensation model of RELAP5/MOD3.1 for passive high-pressure injection system” *Annals of Nuclear Energy* 25(9): 677-688.

LEE, S.I., NO H.C., (1998a) “Gravity driven injection experiments and direct contact condensation regime map for passive high-pressure injection system” *Nuclear Engineering and Design 183 213-234.*

MARSH, C., (2004). “Numerical simulation of flashing flow.” *MEng Thesis*, University of Limerick, Ireland.