Abstract

The method of introducing black liquor into a recovery boiler is critical to a successful operation. The nozzle spray has an effect on the furnace bed size and shape, bed and furnace outlet gas temperatures, fuming rate and carryover. The selection of the type of nozzle and its operating parameters has traditionally been based on a combination of calculations and experience.

To better understand the performance of a particular nozzle, the determination of the droplet size distribution is critical. In the past, this particle size distribution has been determined by physical simulations in a spray booth, using either black liquor or corn syrup. A computer program is being developed to properly simulate the impingement of black liquor on the plate of the nozzle, and the formation of the liquid sheet and subsequent droplets.

This paper will present the initial results of the development of this computer program. The governing parameters will be reviewed and the results of the validation will be shown. Also, the effect of some of the main parameters on the spray pattern, such as jet velocity, will be illustrated.

Introduction

Black liquor droplet size is an important parameter for proper combustion in a chemical recovery boiler. Droplets that are too large do not have sufficient time to dry and partially pyrolyse before reaching the char bed or walls, causing bed instability and possibly a black out. Droplets that are too fine are entrained with the flue gas. This carryover can deposit on the upper furnace tubes, potentially plugging the flue gas passage.

Droplet mean mass diameter is difficult to measure in-situ. Therefore, although the generally accepted optimum droplet mass mean diameter is approximately 3 mm [1], this has not been validated by extensive field testing. A computer code that could accurately predict black liquor spray flow characteristics and droplet size distribution based on specified operating conditions and nozzle configuration would be invaluable.

From a practical point of view, this computer code would have three main functions:

- to develop an improved nozzle based on understanding the effect of each change in nozzle configuration on spray characteristics.
- given the operating parameters of a particular chemical recovery boiler operation, to predict the black liquor droplet size using a specific nozzle or conversely, to determine the optimum nozzle for the specific operation.
- to provide input into boiler computational fluid dynamics (CFD) to improve predictions of burn out, carryover, and combustion characteristics, resulting in better assessment of the effects of fuel changes on operating recovery boilers. Current CFD modeling cannot accurately predict droplet formation.

In September 2000, work was initiated to develop such a code, entitled BLSpray.

In the past, attempts have been made to empirically measure droplet sizes in a laboratory. Since properties of corn syrup at room temperature are similar to those of black liquor at firing temperature, the spray characteristics of corn syrup were studied in a laboratory experiment. This paper will discuss validation of the model using these experimental results.

Numerical Methods

BLSpray is based on a 3D model of liquid flow which predicts liquid free-surface evolution and possible breakup following the fluid flow into the nozzle. The model is an Eulerian fixed-grid algorithm, utilising a volume tracking approach to track free-surface deformation. The model is an extension of the one previously developed to track free surface deformation of a droplet during its impact on a flat substrate [2] and a substrate of an arbitrary shape [3]. The airflow shear stress on the free surface, the turbulence effects of the airflow, and the effects of flow velocity fluctuations are considered in the model.

The full Navier-Stokes equations in a 3D Cartesian coordinate system are solved using a finite volume scheme. The algorithm is currently written for incompressible flows with constant properties. The surface profile of the deforming interface is defined using the 'fractional volume of fluid' scheme. In this method, a scalar function f is defined whose value is equal to the fractional volume of the cell occupied by the fluid. f is assumed to be unity when a cell is fully occupied by the fluid and zero for an empty cell. Cells with values of $0<f<1$ contain a free surface.

In the presence of a solid object (nozzle body), computations of the velocity field have to account for the presence of an irregularly shaped obstacle on which the relevant boundary conditions have to be applied. We treat the solid region of the domain using a modified version of the fixed velocity method [3]. In this approach, a volume fraction Θ is defined whose value is equal to one in the fluid (liquid or gas) and zero in the solid. In general, however, obstacle boundaries arbitrarily snake through the computational mesh, cutting through cells. This gives rise to Θ values in the range $0<\Theta<1$, which is necessary to avoid a 'stair-step' model of a curved interior obstacle boundary. Cells with values of $0<\Theta<1$ are termed 'partial flow cells' because a portion Θ of their finite volume is open to flow and the remaining portion