A method for the continuous fluid sheet formation and its breakup is introduced. In the present model, instead of a continuous liquid jet, streams of droplets impinge on a splash plate and breakup into smaller droplets. Droplets diameter and velocity distributions after impingement are determined based on liquid sheet thickness and velocity obtained from a theory for oblique jet impingement on a plate. Kiva code is modified to carry out the simulation. The result is compared to the experimental data and droplet size distribution across the spray is studied. The effect of different parameters on the final size and velocity distribution is investigated using the developed numerical method.
Introduction

Splash plate nozzles are commonly used in recovery boilers in the pulp and paper industry. These nozzles comprise of a flat plate which is fixed to the outside of a nozzle at an angle between 35 and 55 degrees to the flow. In this type of nozzle, flow stream impinges on the plate forming a sheet of liquid, which consequently breaks to form a spray of droplets. The combination of nozzle angle and plate angle determine the initial trajectory of the flow into the furnace and govern the contact between the fuel and the combustion air. Spray droplet size distribution is a key variable in controlling the boiler performance, namely, droplet combustion, char bed formation, smelt reduction and carryover. Fuel sprayed into the recovery furnace should ideally form droplets small enough to dry and partially pyrolyze before reaching the char bed, but large enough to avoid being entrained in the furnace gas flow.

Proper design of the spray nozzle is of outmost importance in the furnace performance. However, due to the lack of information on the spray size distribution for each nozzle used in the actual furnace, no clear knowledge on the proper size distribution exists. The present practice is a trial and error one, in which nozzles are designed and tested inside the furnace. This practice, has not allowed for a good understanding of what is a proper size distribution, since nozzles are never tested for the size they generate prior to use. Since in furnace experiments are difficult, there is a need to develop accurate models to study various injection strategies, design nozzles that produce the required spray droplet size and velocity distribution, optimize nozzle designs, and finally, study combustion efficiencies.

Currently, there are two methods for predicting the droplet sizes that are generated by splash plate nozzles. One is based on empirically developed correlations, and the other is based on numerical modeling. The empirical models use experimental results in cold conditions, but they are used for the hot furnace conditions (Empie et al. [1], Kankkunen and Helpio [2]). The only available model which uses the basic equations of fluid dynamics to solve for the flow distribution out of a nozzle and can predict the droplet sizes is that of Levesque et al. [3]. However, their model does not account for the effect of air flow on the disintegration process and it needs to be extended to high temperature conditions to predict actual furnace environment.

The objective of this project is to develop a numerical model that can be used to predict the final droplet size and velocity distributions. The method involves discrete phase method along with a sheet formation (on the plate) model. It is fast, accurate and practical and can be used to study the effect of injection conditions, fluid properties, surrounding air, temperature and different nozzle configurations on the droplet size distribution. Kiva code is used as the base code and it is modified to enhance jet impingement on the plate.

Methodology

In the splash plate nozzle, as explained, a continuous liquid jet impinges on a surface and spreads radially forming a liquid sheet. Later, this liquid sheet breaks into small droplets forming the spray. In this research, instead of a continuous liquid jet, streams of droplets are impinged onto the plate, which spread radially and later breakup to form the spray. The final outcome of the two processes (continuous sheet formation and breakup versus discrete droplet injection) will be the same if a proper model is used for the breakup of the droplets after impingement on the splash plate (e.g. a wall model).

Figure 1 shows the schematic configuration of this concept. Droplets are injected from the nozzle. A wall sub-model breaks droplets to smaller drops upon impinging on the plate. Diameter and velocity of produced droplets represents fluid sheet thickness and velocity on the plate. As droplets move further downstream, they may break up by aerodynamic forces. Figure 2 shows the top view of a spray, in which a parent droplet breaks into two or more smaller droplets.

The wall model then serves as an input for a numerical code. In the code, the droplet dynamics is calculated using a spray model, which is based on a droplet probability distribution function, which has eleven independent variables. These are the three components of the droplet position, the three velocity components, droplet radius, temperature, time, distortion of the droplet from sphericity, and its time rate of change.

Wall Model

The wall model which is used here was proposed initially by Inamura et al. [4] and it is for oblique impingement of a viscous jet onto a plate.

Consider a nozzle with a radius of $a$ and a plate length of $l$, and angle of $\theta$. The flow velocity exiting the nozzle is $U_o$ and the fluid properties are $\rho$, $\mu$, and $\sigma$, corresponding to density, dynamic viscosity, and the coefficient of surface tension, respectively. Fluid jet impinges on the plate and forms sheet. The sheet thickness distribution at the tip of the plate is calculated by solving boundary layer equations on the plate:

$$h_{\phi} = \frac{1}{2r_{\phi}} A B_2 + 1.79 \sqrt{r_{\phi}}$$

(1)

or

$$h_{\phi} = \frac{0.642 A B_2}{r_{\phi}} + \frac{5.03r_{\phi}}{A B_2}$$

(2)
Where

\[
 r_\phi^* = \frac{r_\phi}{a} Re^{-\frac{1}{3}}
\]

and

\[
 h_\phi^* = \frac{h_\phi}{a} Re^{\frac{1}{2}}
\]

where

\[
 Re = \frac{\pi \rho_a U_j}{\mu}
\]

\[
 A = \frac{\sin \theta}{\sin \phi^2 + \cos \phi^2 \sin \theta^2}
\]

\[
 B = (- \cos \theta) \left( \pm \frac{\sin \theta^2}{\tan \phi^2 + \sin \theta^2} \right) + \sqrt{1 - \frac{\cos \theta^2 \tan \phi^2}{\tan \phi^2 + \sin \theta^2}}
\]

The selection between Eqs. (1) and (2) depends on whether the plate tip is before or after the fully developed boundary layer on the plate. The location of the fully developed boundary layer is determined by boundary layer equations on the plate.

If \( r_\phi \leq r_{\phi_0} \) (the location of fully developed boundary layer on the plate) equation (1) is used and if not equation (2) is used. The velocity at the tip of the plate is then calculated from the following:

\[
 U_s = U_j \quad (3)
\]

or

\[
 \frac{U_\phi}{U_j} = \frac{1}{0.8998 + 7.042 A^{-2} B^{-4} r_{\phi}^3} \quad (4)
\]

Equation (3) is used along with (1) and equation (4) is coupled with (2). The calculated sheet thickness and velocity distribution are served as inputs for the Lagrangian code. The thickness and the velocity of the sheet are associated with the parcels of droplets on the plate as following:

\[
 d_{\text{parcel}} = h_{\text{sheet}} \quad (5)
\]

\[
 U_{\text{parcel}} = U_{\text{sheet}} \quad (6)
\]

Equations (5) and (6) are applied in every angle with respect to the sheet centerline.

**Modifications to Lagrangian Code**

KIVA code is used as our base code, though it has been modified to simulate splash plate atomizers using the wall model explained previously. KIVA is a general code which has been originated to simulate combustion engines; hence, due to its application, it is customized for engine type injectors which typically produce spray in the form of discrete droplets right from the injection point, as a result; it suits our application and the method which we want to use (discrete droplet injection from the nozzle). It solves the fluid phase in Lagrangian frame work and the surrounding gas phase in Eulerian frame. The governing equations are continuity, momentum and energy and it is capable of simulating turbulent flows. Code calculates the droplet size, velocity and temperature distribution throughout the domain. It solves for droplet injection, breakup, collision, distortion, oscillation and coalescence and it calculates the mass, and momentum and energy transfer between the droplets and surrounding medium.

The code, as explained, is customized for engine simulation and it should be modified accordingly for different simulations. First we have to add a Sub-routine to the main code to enhance droplets on the splash plate according to the wall model. This sub-routine should determine parcel diameter, velocity and number of droplets in it based on the data from wall model at every instance and every location. It also should enforce a new method of breakup. In original code the breakup is defined as changing the parcel diameter, velocity and number of droplets rather than physically breaking it into two or more parcels. The new subroutine breaks parent parcel injected from the nozzle into several new parcels on the plate. Figure 3 shows the result of such a breakup. As a result of this additional breakup method, code should also determine automatically at breakup instance that the droplets are going to break according to which model (original model or newly introduced model); as a result, modifications are performed on the code to handle the breakup method determination.

**Code Setup**

A simple geometry of 9 cubical boxes is used as a chamber for the spray. One of them is specified for the injection point and the splash plate. Figure 4 demonstrates the solution domain and its different regions. Domain size is 100D by 50D by 25D (D is nozzle diameter) and it is large enough to obtain droplet size downstream of the flow. The plate length varies depending on the nozzle, but in the majority of cases it is approximately 6D. Cells are coarse as the Lagrangian frame work does not need fine mesh. The injection time...
is between 200-250 mSec. This time is enough to reach the steady state conditions based on our investigations.

Results
The proposed model should be validated before performing more in depth study. Figure 5 shows the droplet size at the center of the spray as a function of velocity. It is for a nozzle diameter of 2 mm and water as injection fluid. It compares experimental results with numerical data for four different injection velocities. As observed in the graph, numerical predictions are in good agreement with the experimental results. Figure 6 shows the droplet size distribution at the spray cross section downstream of the impingement point. It is for the nozzle diameter of 2 mm and injection velocity of 22 m/s, and the running fluid is water. The droplet size is maximum at the centerline with 320 µm and it decreases toward the edges. It reaches the minimum value of 140 µm at some point and again starts to increase slightly to 180 µm. The size at the center is expected to be maximum since most of mass flux is flowing through the center and the sheet thickness is maximum at the center resulting in larger droplets. In order to model the rim formation the edges of the sheet, the sheet thickness distribution from the wall model is assumed to be periodic. This accounts for the returning fluid from the backward flow in the sheet. The periodic function has results in a rise in droplet size at the edges. Droplet size distribution from the centerline to the edges covers a wide range of sizes and it declines 50% from the maximum to the minimum size.

Figure 7 shows the top and side views of the total velocity. The velocity is maximum on the centerline with 23-25 m/s and it decreases to 10-15 m/s at the edges. The flow slows down toward the downstream and reaches to 19-23 m/s on the centerline at the distance of 100D of the plate tip. The side view of the spray shows that the droplets are dispersing in the plane vertical to the fluid sheet and they lose velocity (momentum) as they detach from the sheet in a random direction. The dispersion intensifies along the spray axis.

The surrounding air is present (desirably or undesirably) in almost all applications for jet impingement nozzles. It can have a significant effect on the final droplet size distribution through its role in breakup and collision processes and consequently affect other processes such as combustion.

The important parameter is the relative velocity between air and the droplets defined as:

\[ U_{relative} = U_s - U_g \]  \hspace{1cm} (7)

A case is examined for the nozzle diameter of 2 mm with injection velocity of 22 m/s. The air velocity is changed from 2 m/s to 22 m/s which result in 0 to 20 m/s relative velocities. Figure 8 shows that as the relative velocity increases (air velocity decreases) the droplet size drops dramatically. The decrease in droplet size is modest till relative velocity of 15 m/s and it starts to drop sharply after that as relative velocity increases, the droplet size decreases. The effect of surrounding air is not only on the droplet size distribution. It also can greatly affect the spray dispersion. Figure 9 shows two simulations with the same injection conditions and fluid properties. The only difference between them is that air is flowing in case (b) while case (a) is in quiescent ambient. In the absence of air flow significant dispersion is observed. This is because many small droplets lose their momentum and detach from the mainstream flow and continue in some random direction depending on local flow stream conditions. In presence of air flow, small droplets which lose their momentum either are removed by air flow or gain momentum and rejoin the main stream. Spray dispersion effect mass distribution in the area of interest and it also change the way which different sprays interact with each other (next section) and its outcome in term of total droplet size distribution.

Conclusion
A new method of numerical simulation is introduced to simulate splash plate atomizers. It proves to be fast and reliable by avoiding heavy computation necessary for two phase flows. This method uses Lagrangian frame work for the fluid phase and Eulerian frame for the surrounding medium and it substitutes continuous sheet of fluid with discrete droplets which are representative of the sheet by their diameter as the sheet thickness and their velocity as the sheet local velocity.

Mean droplet sizes obtained from the code is compared with those of experiments for different injection velocities. The comparison shows that the discrete phase simulation overall gives accurate results. The accuracy is improving significantly toward the larger nozzles and it is due to the fact that the viscous sheet model is more accurate for large scale nozzles.

The droplet size and velocity distributions are obtained from the code. The result for the droplet size shows that the maximum size is at the centerline of the spray, but there are cases (which are not included in this paper) in which the droplet size at the edges are larger than the center line. It is due to the fact that in those cases the sheet is thicker at the edges (on the plate) which will result in larger droplets. The minimum size usually occurs somewhere between the centerline and the edge and can be as half as the maximum size. This amount of variation can greatly affect the combustion process in the furnace. The maximum velocity for the droplets occurs at the center and it decreases toward the edges. But in addition to that, there are two other things to notice: The first one is that the spray velocity is decreasing generally toward the downstream flow. The
reason is the droplets are in contact with quiescent medium. The other thing to notice is the spray dispersion. There are many small droplets with low velocities which are going in random directions and the situation grows toward the flow downstream.

One of the determining parameters of spray dispersion is the air flow on the spray. The simulation using different air flows on the spray shows that co-flow air stream will reduce dispersion significantly, as the low speed droplets will be removed by air flow. If there is other type of air flows such as counter-flow or even vertical to the spray plane, the situation will be different and probably more dispersion will be observed. The air flow has other effect than dispersion and that is its effect on the droplet size. Results show that increasing the relative velocity between air and droplets will decrease the size. One should be cautious in relating air velocity to the relative velocity. In cases where the sheet velocity is more than the air velocity, increasing the air velocity will decrease the relative velocity and where the air velocity is more than the sheet velocity, increasing the air velocity will increase the relative velocity.

Nomenclature

\[ \rho = \text{density} \]
\[ \mu = \text{viscosity} \]
\[ \Phi = \text{angle in the sheet plane} \]
\[ \theta = \text{nozzle angle} \]
\[ A, B = \text{geometrical parameter} \]
\[ d = \text{diameter} \]
\[ Re = \text{Reynolds number} \]
\[ U = \text{velocity} \]
\[ a = \text{nozzle radius} \]
\[ h = \text{sheet thickness} \]
\[ r_\Phi = \text{distance from impingement point to plate tip} \]

Subscripts

\[ \theta = \text{fully developed boundary layer} \]
\[ \Phi = \text{parameter in the angle in the sheet plane} \]
\[ j = \text{jet} \]
\[ l = \text{liquid} \]
\[ s = \text{sheet} \]

Superscripts

\[ * = \text{non-dimensional} \]

References

**Figure 1.** Schematic of droplet injection from the nozzle. Droplets break into smaller droplets downstream.

**Figure 2.** Top view of the schematic of the droplet breakup upon impact on the splash plate. Velocity components of every droplet are determined by the wall model.

**Figure 3.** Parcel breakup on the plate.
Figure 4. Solution domain.

Figure 5. Numerical and experimental results for nozzle diameter of 2 mm with water as running fluid as a function of injection velocity.

Figure 6. Droplet size distribution at the spray X-section for the nozzle diameter of 2mm and injection velocity of 22 m/s, the running fluid is water at room temperature.
Figure 7. Velocity distribution of the spray for the nozzle diameter of 2mm and injection velocity of 22 m/s, the running fluid is water at room temperature.

Figure 8. Effect of air relative velocity on the droplet size distribution for nozzle diameter of 2mm and injection velocity of 22 m/s, the running fluid is water at room temperature.

Figure 9. Effect of surrounding air on spray dispersion. (a) Without air flow. (b) With air flow.