Mixing and Free Surface Flow in Steelmaking Devices

Abstract

In any steelmaking and casting operation, a proper understanding of the level of mixing phenomena as well as free surface flow in different devices is of immense importance in increasing the efficiency of that operation. Keeping in mind the importance of mixing and free surface flow in steelmaking, efforts have been made in the present work to understand numerically the mixing and free surface flow in various steelmaking devices which can improve the quality of steel.

In Chapter-1, mixing phenomena in a RH process has been studied numerically by solving the Navier Stokes equations along with the species concentration equation in a cartesian coordinate system comprising the geometry of the ladle and the snorkel fitted to it. The solution of the species concentration equation has been utilized to compute the mixing time in the RH ladle under different flow conditions. The numerical procedure and solution algorithm has been first verified by comparing the numerically obtained tracer dispersion curve, with the actual plant measurement, which agrees fairly well with each other. Mixing time for the RH process has been computed for different down-leg snorkel size, snorkel immersion depth (SID) and steel velocity within the down-leg and a non-dimensional mixing time correlation has been developed for the RH ladle taking the above three pertinent input parameters into considerations. The correlated non-dimensional mixing time equation predicts fairly well the computed result as well as the actual mixing time being observed in the plant.

Entrapment of air while the drainage is going on is not a desirable phenomenon. So in all casting processes effort is made to avoid entrapment of slag or air. In the present study (Chapter-2) a new methodology (placement of a drain cover over the drainage point) has been suggested to avoid entrapment of air and at the same time drain the maximum possible amount of steel to the caster. In order to study the effect of drain cover on entrapment of air while draining a cylindrical vessel a two dimensional axi-symmetric numerical simulations have been made using a finite volume method. It employs

unstructured grids with cell wise local refinement and an interface capturing scheme to predict the shape of the free surface of water in a cylindrical vessel, thus simulating the entrapment of air into the drainpipe connected to the vessel. A drain cover was placed on top of the drainpipe to delay the entry of air into the drainpipe, thus enabling more liquid to be drained without the entrapment of air bubble. It was found that an increase of the diameter of the drain cover increases the amount of liquid to be drained out before the air could enter into the drainpipe. It was found that irrespective of the initial bath height, air enters the drainpipe at a particular height of the liquid in the vessel. However, when an initial rotational velocity was imparted to the liquid, the height of liquid when air enters the drainpipe depends on the initial bath height. As the initial bath height increases, air enters the drainpipe at a progressively higher bath height (only with the presence of initial angular velocity). But surprisingly when the drain cover is placed the initial bath height, again, has no effect on the height of the liquid (in the vessel) when air enters the drainpipe even in the presence of a definite initial angular velocity. It was also found that an increased diameter of the drain cover helps to drain more liquid without the entrapment of air bubble in all the cases, with and without the presence of initial angular velocity.

In continuous casting operation, the entrapment of mould flux takes place from the surface wave created in the mould at higher casting speed. In Chapter-3, a two dimensional numerical simulations have been performed using a finite volume method to study the entrapment of air from the meniscus. It employs unstructured grids with cell-wise local refinement and an interface-capturing scheme to predict the shape of the free surface, thus simulating the surface wave, that is created in a mould due to the flow from the submerged entry nozzle (SEN). Simulation has been done for 1:6.25 aspect ratio of the mold having a height of 2m with parallel rectangular ports as well as 15^0 upward and downward ports. It has been found that for low inlet velocity of the SEN (less than 1m/s) the maximum wave amplitude of the surface remains below 12mm and no outside air is entrapped by the wave to form a bubble. However, for high inlet velocity (2m/s or more) there are considerable fluctuations on the free surface and the maximum wave amplitude shoot up beyond 70mm at the start up and slowly falls to about 30mm entrapping air bubbles from the surroundings. The movement of the air bubble within the mold and its rise to the free surface where it subsequently collapses has been captured well in the numerical simulation. The overall shape of the free surface matches well, excepting the initial transience, with that of the experimentally observed free surface, although the free surface never attains a perfect steady shape neither in the experiment nor in the numerical simulation due to its continuous oscillation and breaking.