NUMERICALLY SIMULATING THE SOLIDIFICATION PROCESS OF A MELT CASTING METALLIC FUEL PIN MOLD USING FIDAP

Taide Tan¹, Randy Clarksean¹, Yitung Chen¹, Hsuan-Tsung (Sean) Hsieh¹, and Darrell Pepper²

¹Nevada Center for Advanced Computational Methods (NCACM)
²College of Engineering
University of Nevada, Las Vegas
uuchen@clark.nscee.edu

Incorporating volatile actinides, mainly Americium, into the solidification process for a melt casting metallic fuel pin mold has been a serious problem due to Americium’s high vapor pressure. Any miscalculation of flow velocity, specific heat and conductivity can easily cause the early solidification before the molten metal fills the whole mold or the failure of full solidification. The objective of this paper is to use the computational fluid dynamic model to simulate and analyze the solidification processing that provides the better parameter estimation for the process.

A cylindrical metallic fuel pin mold can be simplified to an axisymmetric geometry, while the applied fluid is a high-temperature melt mixture of Am, Pu, Zr, etc. that is treated as an incompressible Newtonian fluid. Uniform initial filling velocity is employed to represent different filling mechanisms, such as the gravity filling and the pressurized filling. To model the cooling effect, we select two mold materials, copper and glass quartz coupling with various boundary conditions.

Flow velocity, specific heat and conductivity were identified as major factors for solidification of the molten metal fills. Flow condition is evaluated by considering both the Reynolds number and the inner relative roughness of the pin mold. The governing equations for the transient analyses of the filling and the phase change include the Navier-Stokes (momentum) equations, the continuity equation, and the energy equation. Commercial finite element method (FEM) software, Fluid Dynamics Analysis Package (FIDAP), was used for simulating the problem. Since an axisymmetric geometry was assumed, only a cross section of the mold from the centerline to the outer boundary need be modeled. A “mapped” type mesh was used to discretize the geometry in our computational domain. A general-purpose pre-processor, GAMBIT, was used for generating the mesh.

An implicit Euler backward scheme was used to solve the transient equations. A segregated iterative algorithm was used to obtain the solution at each time step using a fixed time step size. The streamline upwinding scheme was used to stabilize high-frequency oscillations caused by the convective nature of the flow. Mixed and penalty interpolation for pressure and velocity with discontinuous pressure across element boundaries were employed respectively and the solutions are compared.

To properly define enthalpy, specific heat and viscosity curves during phase change analysis is critical. The temperature range in which the value of the viscosity drops to fluid viscosity value be defined slightly above the temperature range in which latent heat is released. A pseudo-time stepping strategy was utilized to attain convergence. Simulation results are discussed. The possible errors due to our assumptions, postulates and shortcomings of the software are also analyzed.