International Journal of Chemical, Materials and Biomolecular Sciences
ISSN: 2415-6620
Vol:12, No:2, 2018

Investigation on Mesh Sensitivity of a Transient Model for Nozzle Clogging

H. Barati, M. Wu, A. Kharicha, A. Ludwig

Abstract**—**A transient model for nozzle clogging has been developed and successfully validated against a laboratory experiment. Key steps of clogging are considered: transport of particles by turbulent flow towards the nozzle wall; interactions between fluid flow and nozzle wall, and the adhesion of the particle on the wall; the growth of the clog layer and its interaction with the flow. The current paper is to investigate the mesh (size and type) sensitivity of the model in both two and three dimensions. It is found that the algorithm for clog growth alone excluding the flow effect is insensitive to the mesh type and size, but the calculation including flow becomes sensitive to the mesh quality. The use of 2D meshes leads to overestimation of the clog growth because the 3D nature of flow in the boundary layer cannot be properly solved by 2D calculation. 3D simulation with tetrahedron mesh can also lead to an error estimation of the clog growth. A mesh-independent result can be achieved with hexahedral mesh, or at least with triangular prism (inflation layer) for near-wall regions.

Keywords**—**Clogging, nozzle, numerical model, simulation.

I. Introduction

NOZZLE clogging is an important issue of many scientific and engineering branches such as automotive industry, food productions, pharmaceutical industries, and steel continuous casting [1]. In steel metallurgy, when the liquid melt is conducted through a refractory tube, called as submerged entry nozzle (SEN), into the casting machine, the nonmetallic inclusions (NMIs) as solid suspension particles in the steel melt can be transported towards the SEN wall, leading to clogging of the SEN [2]. Due to the high temperature, opaque nature of nozzle and steel melt, experimental study of clogging is not possible. Therefore, numerical modeling seems to be a unique method to study clogging and related phenomena.

There have been various modeling attempts for the clogging of SEN, but most researchers have focused on the first step of clogging, i.e. the melt flow and particle transport, while the phenomena following particle deposition have been largely ignored. For example, Mohammadi et al. [3] studied modeling features regarding turbulence and correlated them to particle behavior inside the nozzle and clogging. Long et al. [4] considered effects of nozzle diameter, mass flow rate, and size of particle on particle deposition on the SEN wall. Ni et al. [5] tracked particles from different injection positions, and Gutiérrez et al. [6] studied the forces acting on particles by the melt flow. All those models/researches have assumed a ‘clean’ nozzle wall, and no narrowing of the nozzle section due to the growth of clog is considered. Therefore, none of the previous models have properly dealt with clogging after the deposition of the particles on the SEN wall. Although one can to some extent investigate the effect of the partially-clogged section (by manually adapting the domain geometry) on the flow [2], [7], the transient growth of the clog layer by continuous deposition of NMIs and its interaction with the flow are largely ignored. In a recent publication of the authors [8], a transient model is developed and validated against a laboratory experiment. The results show that clogging is a transient process including the initial coverage of the nozzle wall with deposited particles, the evolution of a bulged clog front, and then the development of a branched structure; clogging is a stochastic and self-accelerating process.

To extend this model to other operating conditions at the large scale, the influence of numerical parameters on the calculation accuracy needs to be carefully considered. This paper presents a numerical study on the mesh (size and type) sensitivity of the model in both two and three dimensions.

II. Numerical Model in Brief

An Eulerian-Lagrangian model [8] is adopted. The flow of the fluid as the primary continuous medium is described by the conservation equations of mass and momentum along with turbulence equations. Here, shear-stress transport (SST) k-ω model is used because it effectively blends the robust and accurate formulation of the k-ω model in the near-wall region with the free-stream independence of the k-ε model in the far field [9].

The solid suspensions are assumed as spherical particles. A force balance of buoyancy, drag, lift, virtual mass, and pressure gradient controls the motion of them. A stochastic model [10] is implemented in the wall boundary cells to mimic particle motion in near wall region. Once a particle meets the wall, it is assumed to be attached on the nozzle wall permanently. This assumption is considered because the capillary force, also termed adhesion force, is the dominant force among all forces acting on the particle [11]-[13].

Fig. 1 shows the clogging procedure schematically. In the early stage, Figs. 1 (a) and (b), particle deposition leads to change in wall roughness. This stage lasts until the roughness height is larger than half of boundary cell thickness. Thereafter, the boundary cell is converted into a partially

H. Barati is Ph.D. candidate at Montanuniversitaet Leoben and researcher at K1-MET, Franz-Josef Str. 18, 8700 Leoben, Austria (corresponding author, phone: +43 3842-402-3119; e-mail: hadi.barati@unileoben.ac.at).

M. Wu is associate professor at Montanuniversitaet Leoben, Franz-Josef Str. 18, 8700 Leoben, Austria (e-mail: menghuai.wu@unileoben.ac.at).

A. Kharicha is Ph.D at Montanuniversitaet Leoben, Franz-Josef Str. 18, 8700 Leoben, Austria (e-mail: abdellah.kharicha@unileoben.ac.at).

A. Ludwig is professor at Montanuniversitaet Leoben, Franz-Josef Str. 18, 8700 Leoben, Austria (e-mail: andreas.ludwig@unileoben.ac.at).

clogged call, i.e. the cell is filled with clog material (porous medium), marked in gray in Fig. 1 (c). The part covered by the clog material is quantified as volume fraction of deposited particle (f_p). Further deposition results in a full occupation of the cell by particles, as indicated by line pattern in Fig. 1 (d). To take into account the effects of clog growth on the fluid flow, Darcy source terms considering the permeability of the clog material are applied for the clog region. More details about the model can be found in [8].

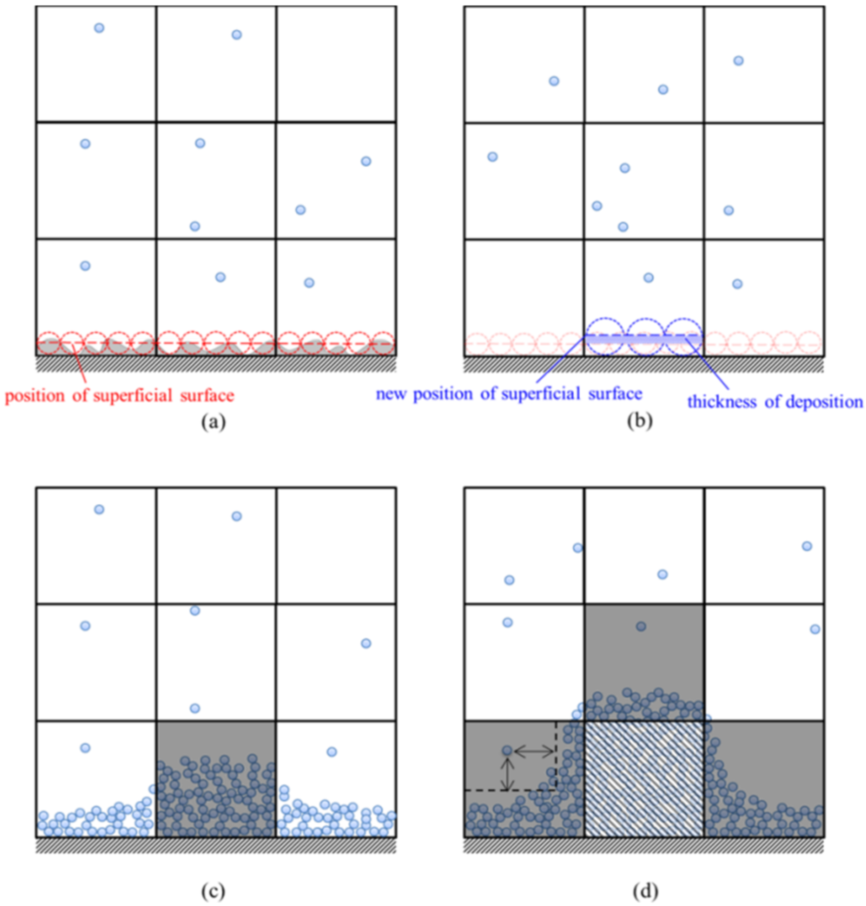

Fig. 1 Schematic illustration of clogging procedure: (a) initial wall roughness considered as uniform sand-grain roughness, (b) enhanced wall roughness due to the particle deposition, (c) formation of porous region of clog, and (d) clog growth

It is well known that many computational fluid mechanics (CFD) issues depend on mesh quality. During the calculation of nozzle clogging, both clog growth and the flow would be sensitive to the mesh quality. In order to investigate the influence of mesh quality on the clog growth individually, we need to define an ideal case of the 'pure' clog growth, as shown in Fig. 2 (a), where no flow is involved but particles move automatically towards the nozzle wall. A uniform distribution of particles is injected at the top surface of a simple computational domain, and particles move towards the wall until they meet the wall or clog. A bunch of spherical particles with diameter of 10 μm are in injected and move to the wall successively. All bunches are identical, and 45 million particles are injected in total.

To study the effects of the mesh quality on the clogging including flow effect, a simple condition is considered for the computational domain. A fully developed turbulent flow between two parallel plates is simulated. The gap between two plates and the mean velocity of the fluid are 5 mm and 5 m/s, respectively. These values are extracted from one typical metallurgical application, i.e. SEN clogging during continuous casting (flow through the gap between stopper and SEN). The domain and boundary conditions are schematically shown in Fig. 2 (b). On the top surface, a fully developed flow profile is assigned to the inlet and pressure-outlet is imposed for the outlet. The front and back surfaces are supposed to be symmetry planes. Non-slip boundary condition is applied on the wall. For better visualization of clog growth, a certain area of the wall is defined for particle attachment, as indicated in Fig 2 (b) and in other areas rebound condition is considered. Due to the symmetrical condition, only half of the geometry is simulated. Particles with diameter of 10 μm are injected uniformly from the inlet. The injection rate of particles is 20 million particles per second. The material of the particles is alumina. The simulations are performed in both 2D and 3D. Different types of mesh are taken into account (Fig. 2 (c)): hexahedron, tetrahedron, and inflation tetrahedron. The term 'inflation' is used in ANSYS-FLUENT meshing tool to generate triangular prism cells in the near-wall region, while in the inside parts tetrahedron is used. The thickness of the inflation layer is 1 mm. The equations are numerically solved using commercial CFD code ANSYS-FLUENT with extended user-defined functions (UDFs) for considering the particle deposition and the growth of clog.

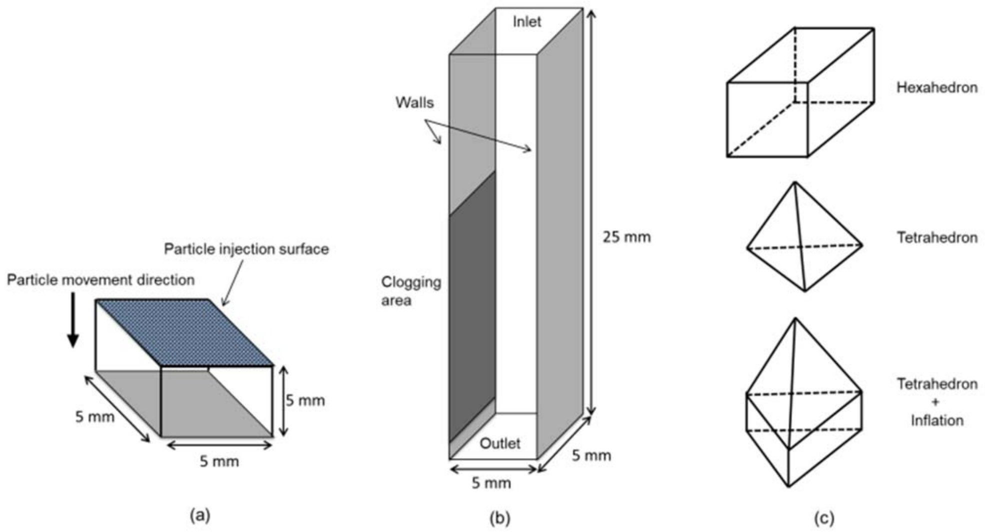

Fig. 2 Domain and boundary conditions for the simulation of clog growth: (a) without and (b) with flow. (c) Three types of mesh are used in the current study. Deposition area on the wall for parallel plates is marked in darker color

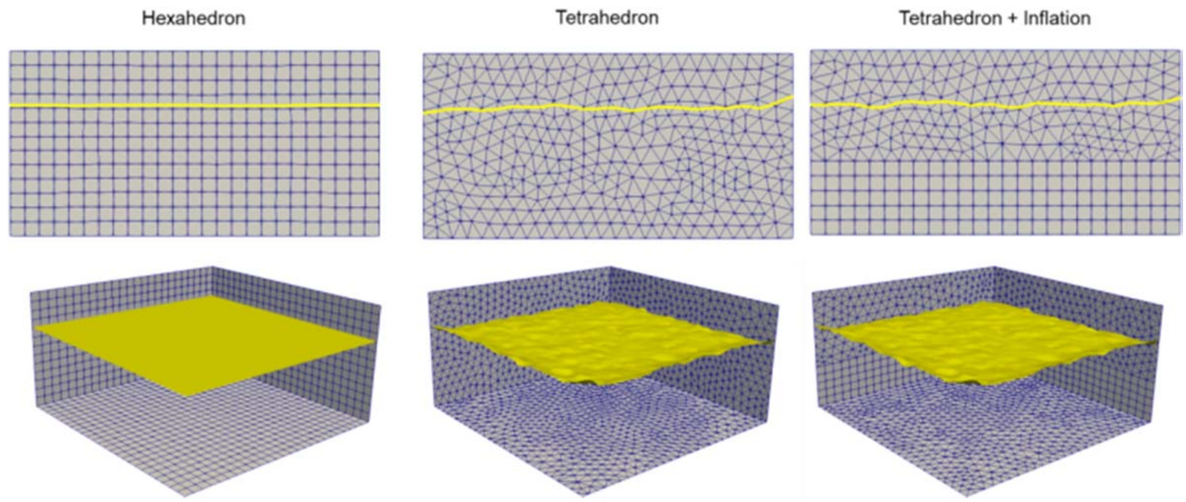

Fig. 3 Comparison of the positions of clog front (yellow iso-line or iso-surface) after deposition of all particles for 3 different mesh types in two dimensions (upper) and three dimensions (lower)

III. Results and Discussion

Results of clog growth without consideration of the fluid flow are shown in Fig. 3. The clog front is shown by an iso-line (2D) or iso-surface (3D) where the volume fraction of deposited particle is equal to half of a predefined average value ($\bar{f}_{\mathrm{p}}$). In the current paper, $\bar{f}_{\mathrm{p}}$ is 0.55. For better visualization of 3D results, the mesh is plotted on the domain walls. The clog fronts after deposition of all particles for all three mesh types are almost at same position in both 2D and 3D. One can conclude that the calculation of clog growth is independent of the mesh type and dimensions.

Calculations with coarse mesh (doubled the cell size) are made. The results show that the average height of clog front changes by 2.8%, 2.7%, and 2.0% in 2D for rectangular, triangle and mixed mesh types; and 3.1%, 2.8%, and 1.8% in 3D for hexahedron, tetrahedron, and inflation tetrahedron, respectively. Therefore, mesh size does not have a significant influence on the clog growth as well.

To study the clog growth with fluid flow, we should firstly ensure the mesh independency of the flow calculation. In this regard, different uniform meshes are generated with different cell size: 0.10, 0.15, 0.20, 0.25, and 0.30 mm. The results indicate that, in 3D calculation by decreasing the mesh size from 0.20 to 0.15 mm, the average change in velocity and turbulent kinetic energy profiles are < 1% and < 2%, respectively, for three mesh types. Accordingly, mesh size of 0.20 mm is selected for the simulations. The velocity profile and turbulence kinetic energy profile are shown in Fig. 4 for three mesh types. For all mesh types, the velocity profiles are identical. The turbulent kinetic energy seems to be significantly underestimated by the tetrahedron mesh.

Transient clog growth including fluid flow and their interactions is shown in Fig. 5. In both 2D and 3D cases, clog growth starts from the upper edge of deposition area, depicted in Fig. 2 (b), and then gradually extends in lower parts. No clog front is visible at 6 s for the triangle/tetrahedron mesh. This does not mean that there is no particle deposition for this case, but the volume fraction of deposited particle in the surface cell is lower than the predefined average value. In 2D cases (at 18 s), three mash types show similar clog front, while in 3D cases results seem quite different.

For the 3D cases, when particle deposition at a point is higher, a bulge forms; then the fluid can flow around the bulge (on top and sides). While for the 2D case, the fluid can only flow on the top of the bulge. Since clogging is a self-accelerating process [8], the bulge extends in different directions in 3D cases. In fact, particles can meet the bulge from different sides.

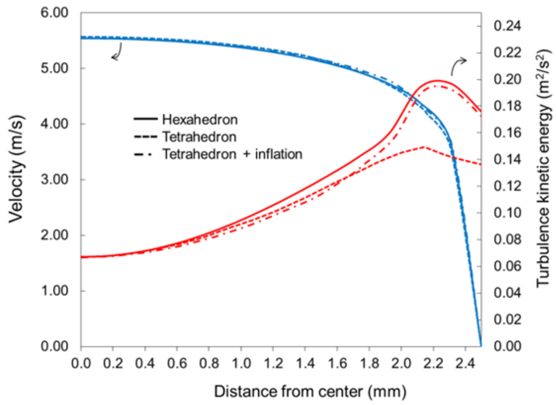

Fig. 4 Fully developed velocity profile and turbulent kinetic energy with different mesh types. The cell size is 0.20 mm

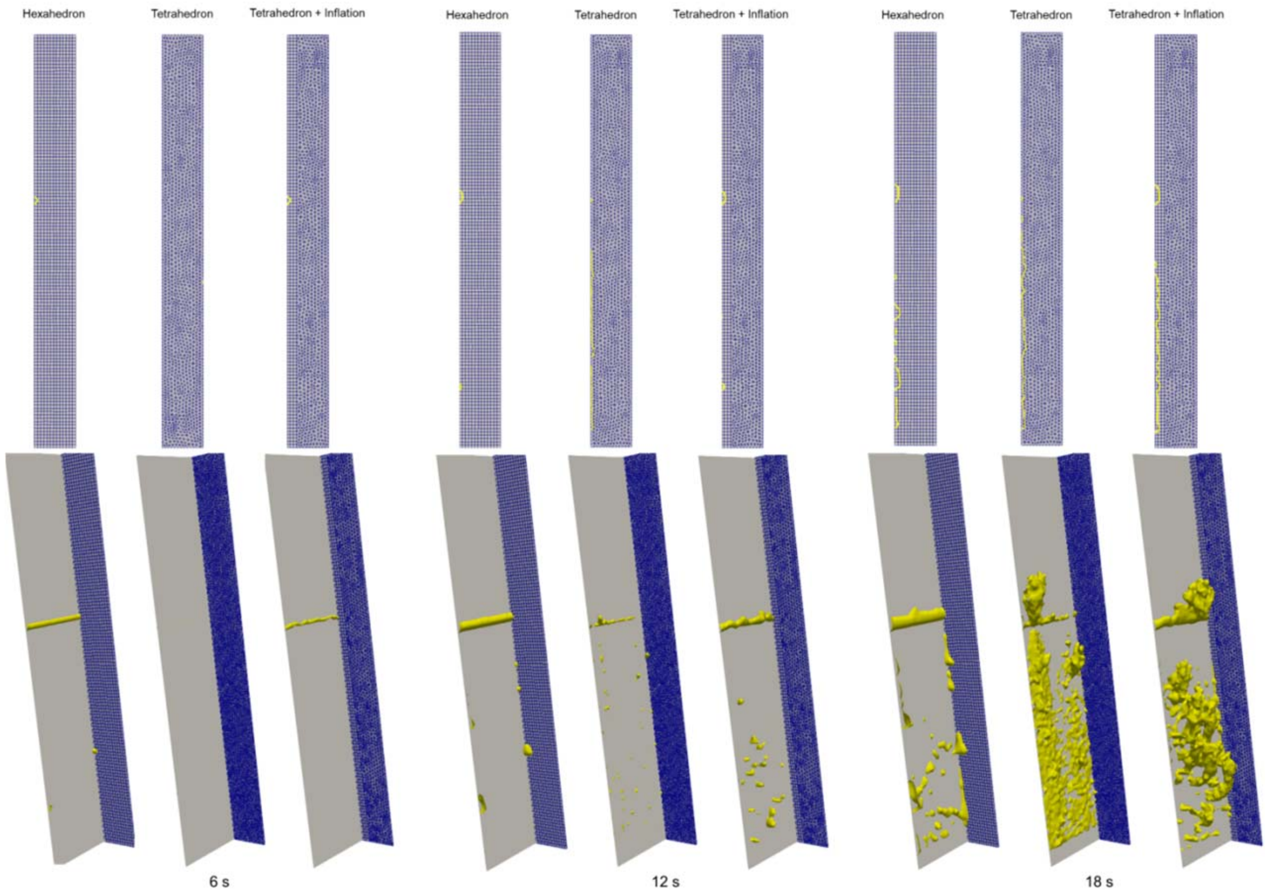

Fig. 5 Transient clog growth for 2D and 3D calculations

Fig. 6 shows total deposit mass as a function of time for three mesh types in 2D and 3D calculations. Fig. 6

quantitatively proves that in 2D condition, three mesh types result in similar deposition mass, as shown in Fig. 5 qualitatively. In inflation part of the 3D mesh, cells are prism with triangular base, and Fig. 6 declares that deposition mass for this mesh is close to 3D hexahedron mesh before 12 s. After this time, clog enters into tetrahedron part and then particle deposition mass increases dramatically. Hence, it can be concluded that inflation part acts like hexahedron mesh.

In Fig. 6 deposition mass of 2D cases generally are higher than that of 3D cases. This is due to the different flow pattern after clog growth in 2D and 3D conditions. In 2D, the fluid can only flow to the top of the bulges, while in 3D it can flow around the bulges. In the other words, the fluid has fewer ways to go around the bulges in 2D than in 3D. Therefore, the probability of reaching of particles the clog surface becomes larger in 2D than in 3D calculations. Among 3D cases, tetrahedron mesh type shows smaller deposition mass. The reason for this difference is lower value of turbulent kinetic energy near the wall for tetrahedron mesh, as depicted in Fig. 4. The lower value of turbulent kinetic energy near the wall results in smaller lateral fluctuation velocity of particles due to the turbulence. Therefore, the chance of particles to reach the wall decreases.

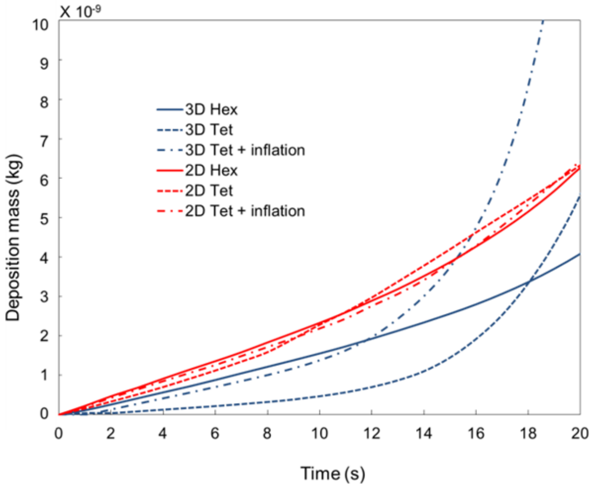

Fig. 6 Total deposition mass as a function of time for different mesh types. To make 2D results compatible with the 3D ones, 5 mm depth is assumed for the 2D condition

IV. Conclusions

Mesh sensitivity of a transient clogging model is numerically studied. Three mesh types are considered: hexahedron, tetrahedron, and inflation tetrahedron. The results show that, for the case excluding flow calculation, the clog growth seems not sensitive to mesh type and size. This point is confirmed in both 2D and 3D calculations. When the flow calculation is (it has to be) included, the simulation in 2D leads to an overestimation of the particle deposition. For the 3D calculation, tetrahedron mesh leads also to unphysically too-high particle deposition, because tetrahedron mesh is generally not suitable for modeling turbulent flow in the near-wall region. A mesh independent of clogging result in the 3D calculation can be achieved for hexahedron mesh, or prism mesh with triangular base (inflation tetrahedron). Therefore, it is recommended that if hexahedron mesh is not possible due to the complexity of the domain, inflation mesh is a proper solution for clogging simulation. The thickness of the inflation layer should be correlated to the expected thickness of the clog.

References

[1] C. Henry, J.-P.P. Minier, and G. Lefèvre, "Towards a description of particulate fouling: From single particle deposition to clogging," *Adv. Colloid Interface Sci.*, vol. 185– 186, pp. 34–76, Dec 2012.
[2] H. Bai, and B.G. Thomas, "Effects of clogging, argon injection, and continuous casting conditions on flow and air aspiration in submerged entry nozzles," *Metall. Mater. Trans. B.*, vol. 32, pp. 707–722, Aug 2001.
[3] M. Mohammadi-Ghaleni, M. Asle Zaeem, J.D. Smith, and R. O'Malley, "Comparison of CFD simulations with experimental measurements of nozzle clogging in continuous casting of steels," *Metall. Mater. Trans. B.*, vol. 47, pp. 3384–3393, Dec 2016.
[4] M. Long, X. Zuo, L. Zhang, and D. Chen, "Kinetic modeling on nozzle clogging during steel billet continuous casting," *ISIJ Int.*, vol. 50, pp.712–720, May 2010.
[5] P. Ni, L. Jonsson, M. Ersson, and P.G. Jönsson, "Turbulent flow phenomena and Ce2O3 behavior during a steel teeming process," *ISIJ Int.*, vol. 53, pp. 792–801, Jan 2013.
[6] E. Gutiérrez, S. Garcia-Hernandez, and J. Barreto, "Mathematical Analysis of the Dynamic Effects on the Deposition of Alumina Inclusions inside the Upper Tundish Nozzle," *ISIJ Int.*, vol. 56, pp. 1394–1403, Aug 2016.
[7] L. Zhang, Y. Wang, and X. Zuo, "Flow transport and inclusion motion in steel continuous-casting mold under submerged entry nozzle clogging condition," *Metall. Mater. Trans. B.*, vol. 39, pp. 534–550, Aug 2008.
[8] H. Barati, M. Wu, A. Kharicha, A. Ludwig, "A Transient Model for Nozzle Clogging," *Powder Technol.*, accepted, 2018.
[9] "ANSYS-FLUENT help," Acad. Res. Release. 14, 2015.
[10] M. Guingo, and J.-P. Minier, "A stochastic model of coherent structures for particle deposition in turbulent flows," *Phys. Fluids.*, vol. 20, p. 53303, May 2008.
[11] F. Heuzeroth, J. Fritzsche, E. Werzner, M.A.A. Mendes, S. Ray, D. Trimis, and U.A. Peuker, "Viscous force - An important parameter for the modeling of deep bed filtration in liquid media," *Powder Technol.*, vol. 283, pp. 190–198, Oct 2015.
[12] K. Sasai, and Y. Mizukami, "Mechanism of alumina adhesion to continuous caster nozzle with reoxidation of molten steel," *ISIJ Int.*, vol. 41, pp. 1331–1339, 2001.
[13] K. Uemura, M. Takahashi, S. Koyama, and M. Nitta, "Filtration mechanism of non-metallic inclusions in steel by ceramic loop filter," *ISIJ Int.*, vol. 32, pp. 150–156, 1992.