Sociedad Mexicana de Ingeniería Mecánica
Fecha de recepción: 05-10-2018
Fecha de aceptación: 06-02-2019

INGENIERÍA MECÁNICA
TECNOLOGÍA Y DESARROLLO
Vol. 6 No. 4 (2019) 151 - 156

Permanent mold filling process modeled by CEL elements

Modelado del proceso de colada en molde permanente mediante elementos CEL

Alan Rodrigo Saucedo Osorno, Edgar Isaac Ramírez Díaz, Osvaldo Ruiz Cervantes
Armando Ortiz Prado
Departamento de Manufactura y Materiales de la Facultad de Ingeniería
Universidad Nacional Autónoma de México Circuito exterior de Ciudad Universitaria
Coyoacán, C.P. 04510, México, CDMX. Tel. 5622-80-57, Fax 5622-80-58

Abstract

Simulation of a permanent open mold filling is presented with the aim to analyze the capacities of CEL Abaqus tool. The fluid behavior is defined by Mie-Grüneisen equations, driven only by body forces in order to identify areas with turbulence and relate them with defects in the solidified part. The filling is achieved using the fluid-solid interaction (FSI). Finally, free surface behavior, droplet formation, velocity vector field, volume distribution within the Eulerian mesh were obtained. It is concluded that the analysis CEL can be an alternative tool to the CFD analysis, since it was able to model the multiphysical phenomenon of casting process, being able to relate the regions with turbulence with the defect zones in the solidified ingot.

Resumen

En este trabajo se presenta la simulación de una colada en una matriz abierta con el objetivo de analizar la capacidad de la herramienta de Abaqus™ CEL para su modelado. El comportamiento del fluido se definió mediante el modelo de Mie-Grüneisen, la única carga aplicada fue la gravedad para así identificar las zonas con turbulencia y relacionarlas con los defectos en la pieza solidificada. El llenado se consiguió mediante el algoritmo contacto general explícito (GC) para lograr la interacción fluido-sólido (FSI). Finalmente se obtuvo el perfil de flujo, comportamiento de superficies libres, formación de gotas, campo de velocidades, fuerzas de reacción, distribución de volumen en el dominio Euleriano. Se concluye que el análisis CEL puede ser una herramienta alternativa a los análisis CFD, ya que fue capaz de modelar el fenómeno multifísico del proceso de colada, pudiendo relacionar las regiones con turbulencia con las zonas con defectos en el lingote solidificado.

Keywords:
Mold filling, Fluid-Solid Interaction (FSI), Coupled Eulerian-Lagrangian analysis (CEL), Finite element

Palabras clave:
Colada, Interacción Fluido-Sólido (FSI), Análisis acoplado Euleriano Lagrangiano (CEL), Elemento Finito

Introduction

In gravity casting processes, any variation from laminar flow has critical implications to solidification and product quality [1] resulting in defects in the solidified part [2], such as pores, inclusions, oxide formations and air and gas trapping [3]. The behavior of fluid flow in real molds is determined experimentally by analyzing the solidified pieces and the filling process observation, which involves costs and material, it could be optimized using specialized software and with the appropriate approach, valid simplifications and a correct interpretation, it will show accurate results in comparison with experimental data [4 - 12]

Since the beginning, simulation of cavities filling has focused on the analysis of a single variable as: speed, pressure, heat transfer, convection, and others, this limitation is mainly due to the model complexity required to study some variables mentioned as another function and limited computing capacity. Nowadays with the advance in the computational modeling, the temperature variation as a function of time and material flow simultaneously analyzed has been achieved [13, 14]. However, this requires extensive programming and it is limited to a specific problem, also demands long time of calculation in order to obtain an acceptable numerical solution. Recently with the multivariate analysis implementation switching several types of coordinates (Eulerian and Lagrangian) in a single model is possible [15].

The CEL method avoids the problem of using a Lagrangian approach for fluids, limited by mesh quality, implementing a traditional Eulerian mesh, allowing material flow without mesh deformation [16]. Some advantages of CEL analysis is to initialize the material as a predefined field in specific sections of the mesh, that are defined with partitions or using a reference part with a specific shape for the material. This requires the use of the Volume Fraction Tool (VFT) that assigns a volume from 0 to 1 on the Eulerian elements interfered by the elements from reference part and define the initialization field for Eulerian material and create an element set where it is possible to assign any initial condition [15]. In order to ensure a correct FSI is necessary a ratio between Lagrangian and Eulerian elements of one Lagrangian per minimum 3 and maximum 5 Eulerian elements [17]. In addition, the pre-programmed model used for the proposed test is the Newtonian behavior with semi-compressible type described by Mie-Grüneisen equation of state (EOS) with li-

near approximation in the pressure made by Hugoniot which is [18]:

$$p = \frac{\rho_0 c_0^2 \eta}{(1 - s\eta)^2}\left(1 - \frac{\Gamma_0 \eta}{2}\right) + \Gamma_0 \rho_0 E_m \quad (1)$$

Where p is the magnitude of the system pressure, ρ_0 initial density, c_0 is the sound wave speed in the fluid material at the reference state, η nominal volumetric deformation due to compression, Γ_0 Grüneisen constant at reference state, E_m energy per mass unit and ***s*** is the line slope that shows the relation between the particle velocity and the applied velocity on the fluid.

The product $c_0^2\ \eta$ of **eq. 1** approximates to compressible modulus, responsible for modeling the material under impact, valid only when η is small; these systems present a critical strain and compressibility. This point sets the maximal stress state, exceeding this value appears negative sound speed, this is interpreted such as the material makes sound or like a numerical mistake in the solution, this depends entirely on the phenomenon.

Methods

Geometry description

The Lagrangian part in the simulation approach is a rectangular cast iron mold used to solidify 350 gr Cu alloy ingots [19] its dimensions are reported in Table 1.

Table 1 Dimensions used to build the cavity.

Dimension	Value mm
X	88
Y	143
Z	7
Radius edges	12
Sprue radius	4

The VFT reference part is a cylinder with 30 mm radius and 17.2 mm height obtained to ensure the volume of the molten metal. In addition, a funnel is needed for channeling the fluid to the sprue of the mold which has a total height of 42 mm, smaller radius of 4.36 mm and two rounds of 10 mm to ensure the relation between Eulerian and Lagrangian elements, the first is located at 13 mm and the second at 38 mm from the upper edge.

These solid pieces were built as shells, defined with rigid body constraint. Finally, the domain is a simple box 30 mm height, 11 mm width and 75 mm depth.

Material Properties

Casting flow modeling is often performed with water in transparent molds, this is possible since the kinematic viscosity of water is similar to most metals at filling temperature [3] this implies that water properties are acceptable for the model and are shown in Table 2. These properties are used to satisfy the Mie-Grüneisen EOS and Hugoniot approach.

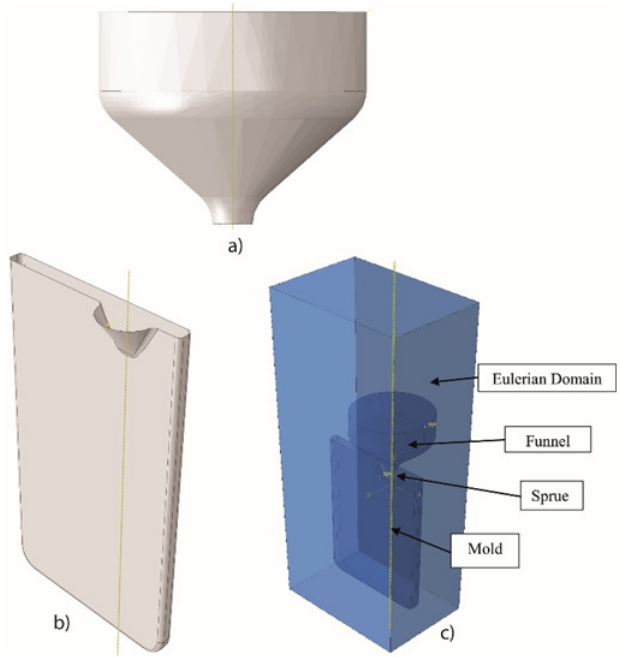

Figure 1 a) Funnel, b) Mold, c) Simulation assembly.

Table 2 Properties for Eulerian material definition [8].

Parameters	Value
ρ_0	998.2 kg/m^3
v	0.001003 N s/m^2
c_0	1500 m/s
s	0
Γ_0	0

Mie Grunëisen equations are used for large deformations in deformable material as a function of internal energy, dynamic viscosity, and speed of sound propagation inside the fluid. Mass conservation was added for each Eulerian domain modeled as the total sum of mass moving through the element faces, equal to zero. Moment conservation applied in the node of both domains (Eulerian and Lagrangian) is considered, by using full sum of liner and angular moment equal to zero.

The main process considered were momentum transfer and viscous collision in Eulerian elements, in order to calculate mass transfer along the Eulerian domain. No heat transfer between solid and fluid domains was modeled.

The fluid was a Newtonian one because the pouring temperature highly above of solidus line. In order to individualize this analysis from other pouring analysis the sound speed propagation c0 through the fluid was used in the Mie-Grunëisen

model. In addition, viscous damping and dynamic viscosity values of water were used. Finally, none solidification effects were considered.

Analysis conditions

The mold filling requires to define dynamic explicit steps on CEL analysis, in that way the numerical solution starts with position and velocity values already known from initial conditions and previous increment to evaluate the values of next increment, this avoids the need to determinate the stiffness matrix and simultaneously allow to restore the calculation from a specific step or increment [20].

One of the major setbacks in the CEL approach is that any material defined by EOS does not accept mass scaling factor, nor can use the velocity scaling because their behavior is dependent on it and result in pressure modification then the computing time could not be accelerated. The only load consider was due to gravity 9.81 m/s^2 in z direction, under this condition, views were obtained when the twentieth of the total set time is reached, therefore 15 identical steps were defined, each one 0.1 s as a result, the resolution obtained was of one visualization per 5e-3 s for a total of 1.5 s simulated to compare with the ingot obtained [11].

The process is constant over time and the only interaction present is FSI so only is needed to define one friction contact interaction as mechanical properties to control viscous fluid behavior [21], this was achieved applying GC algorithm to the entire model. The definition of rigid body (RB) for all solid parts avoids defining specific material or some property, only the inertial behavior of the section [22]. The zone defined as RB becomes into a collection of nodes, or linked elements to a single behavior defined by the user at reference node, the entire collection maintains the relative position, allowing work piece displacement and resultant force calculation [23]. The above definition is applied in all the solid pieces for establishing the desired behavior during testing.

In order to prevent fluid loss mass was necessary to apply constrains on Eulerian domain, by imposing boundary conditions with normal zero velocity to the surface of interest [16] and the Eulerian domain acts as material containment, this was just a precaution to maintain stable energy and mass in the model and does not interfere with the FSI calculation. Also, the molten fluid of the Eulerian domain was considered as isothermal.

The Eulerian material was initialized as a predefined field generated by the VFT using the reference part located in the funnel and was applied as material initialization. Mesh definition on all parts was as follow, the domain used 8 nodes cubic Eulerian element EC3C8R overall size of 3 mm, the reference part was meshed with a discrete rigid element of 4 nodes for 3D models (R3D4) with overall size of 1 mm. The funnel used the same element type that reference part but with a global size of 1.5 mm. The mold used an element type equal to the other solid pieces with a global size of 2 mm.

The size is the result of looking for an optimized ratio of elements between three to five of the Eulerian domains for each of the Lagrangian [17], because if it is followed the relation at its minimum dimension, it would get an element size of 0.66 mm for the domain. In this case the calculation time is approximately one day to obtain the first observable result (10% of the total time of one step), so the size is increased at 1 mm, then it is obtained an advance of 60% of the total time of a step in a day of computation.

Eulerian surface tracking and the assigned material volume recognition algorithm can cause "runoff" at edges. The solution method is to get a finer mesh so that there are no Eulerian elements that interact partially with the Lagrangian, but the number of elements impacts the computing time, so the option was to remove edges where there were severe reductions and elements that converge to a single node.

Results and discussion

The velocity profile shows a free flow and acceleration at the funnel reduction, **Fig. 2**, while the distribution of VE shows the formation of vacuum which is confined at the time the fluid contacts itself at the end of the funnel (upper figure), simultaneously the velocity profile is uniform over the solid surface, this suggests that fluid behavior is acceptable during FSI simulation, the vacuum generated by the acceleration, causes no visible effect on the velocity profile, on the other hand the vacuum movement suggests being calculated as an independent entity in the model (lower figure).

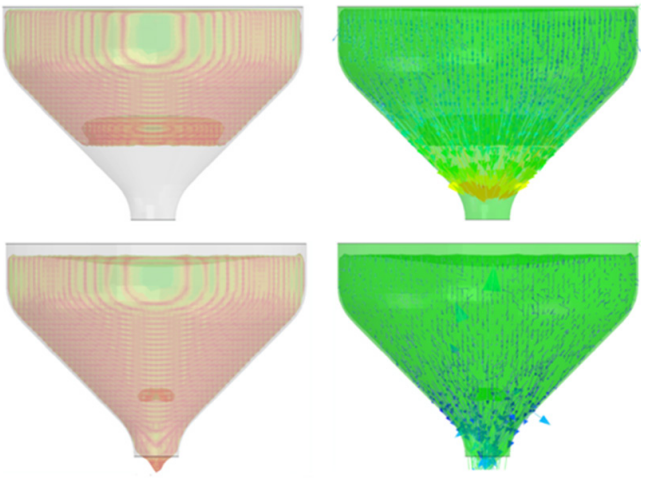

Figure 2 Left column shows the distribution of VE, Right column shows the fluid velocity profile.

The cavity formation in the **Fig. 2** (left), is filled with void, which is applied in the software with the purpose of exchanging defined material through the Eulerian mesh.

This cavity confined void material due to fluid flow, then the "bubble" travels upward and the volume decreases due to the fluid pressure, when the fluid volume decrease the bubble expands and as a result a meniscus appears on the free surfa-

ce, when the surface tension of the fluid defined not support the thrust of the material void, the fluid is projected, forming a column with a drop at the top and a cone at the free surface, and finally the fluid driven by the "bubble" falls by gravity as two drops on the top, as the cone superficial area increases and appears sloshing.

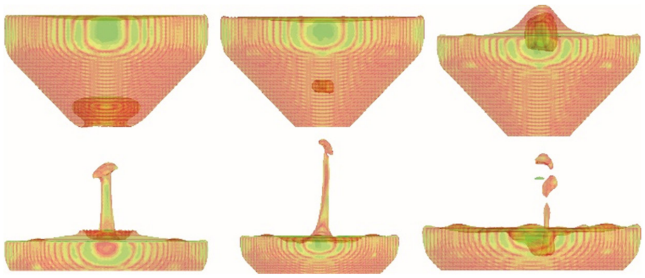

Figure 3 Void material behavior in the funnel.

The results validation was done by comparison with high-speed videos (5e3 and 10e3 frames per second (fps)) with resolution of 1e-3 and 2e-3 s.

The **Fig. 4a**, shows the formation of a fluid meniscus, **Fig. 4b** shows a column of fluid accumulation at the top forming a droplet and two different regions, one with waves near the cone under the column of fluid and other quit zone beyond first one, **Fig. 4c** shows two droplets due to the inertia of each fluid cluster, each one showing different response to the gravity, this indicates that the calculation is very close to reality results.

a) b) c)

Figure 4 Fluid under impact, a) and b) was taken at 5000 fps [13], and c) at 10000 fps [15].

The measurement of the reaction force at the funnel and the mold show that the value is constant, except for short periods of time, which are presented at the same moment for both bodies $t_2 = 0.1$ s and $t_3 = 0.2$ s, the first one corresponds at the moment that a bubble gets to de free surface of the fluid and forms a meniscus and the second one at the moment when the flux change of direction and started the recirculation at the bottom of the mold. For the funnel a first peak is presented at $t_1 = 0.055$ s, it corresponds when the fluid passed through the reduction and a bubble precipitates into the fluid (**Fig. 5**).

The first simulation step shows the first contact with the mold, the spread of the fluid on the solid walls and splash effect by collision with the sprue, and this effect is covering more area while fluid flow (**Figs. 6a** and **6b**). During the second step the fluid shows an increase in volume confined into the mold, the splatter profile remains constant, accumulation at the bottom and a right to left flow tendency are evident (**Fig. 6c**).

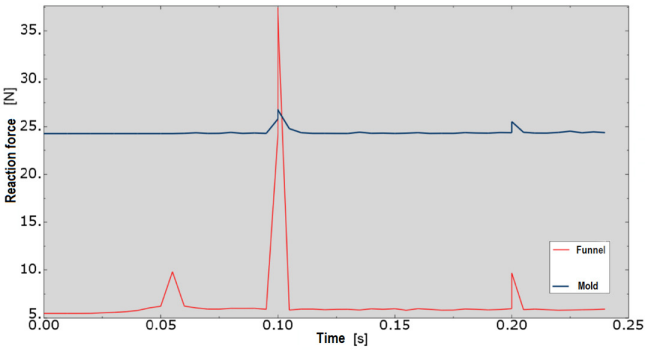

Figure 5 Reaction force of the funnel and mold during filling.

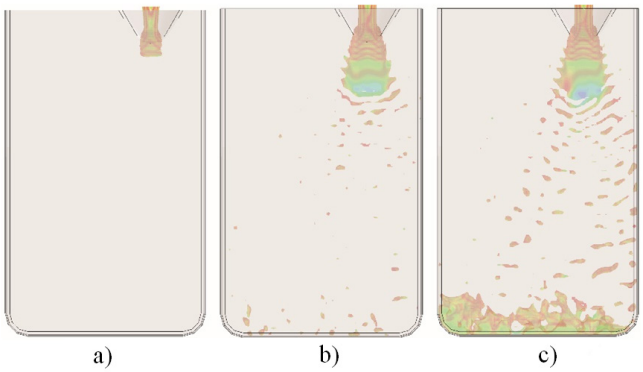

Figure 6 Mold filling at first step a) t = 0.06 [s], b) t = 0.12 [s], c) t = 0.24 [s]

Verification of fluid dynamics is driven analyzing velocity vectors. In the Fig. 7a the sprue displays properly the velocity vectors, these profile follows the solid surface at values between 1 - 5.67 m/s, in Fig. 7b vectors and fluid rise in the cavity at any time has a profile compatible with the distribution of volume and the direction of gravity, and Fig. 7c, speed and flow remains constant and it is only affected by the free surface elevation at values between 0.63 - 1.89 m/s at the bottom of the mold.

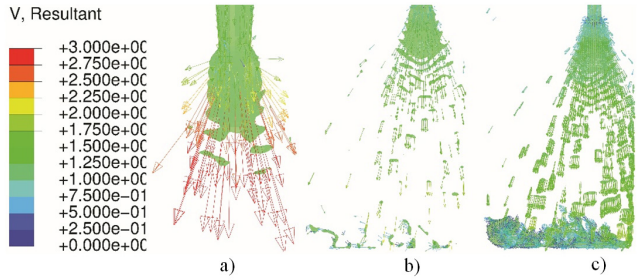

Figure 7 Initial velocity profile, a) t = 0.06 s, b) t = 0.12 s, c) t = 0.24 s

The progress of simulation showed a cyclical pattern, fluid accumulates on the left side, increasing the height and collapses toward the center, splash was produced on the right side and restart the cycle, at the bottom on the left appear two directions of flow facing each other, as a result at the center a vortex was created. Fluid follow the surface defined properly by the solid (Fig. 8).

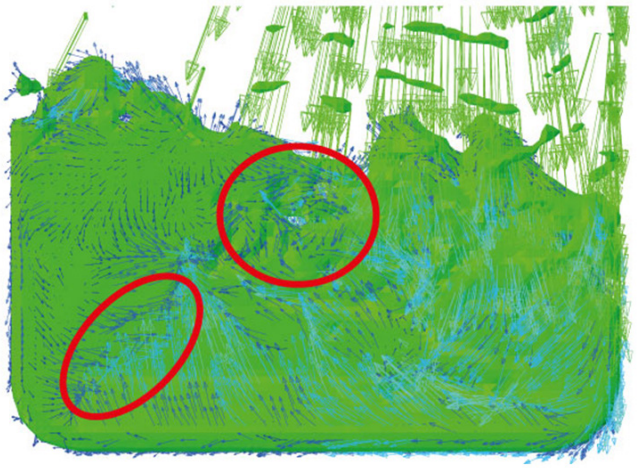

Figure 8 Velocity profile at t = 0.655 s

In this paper the turbulence was defined as any area with high velocity vector direction changing randomly, volume per eulerian element changes and circulations of fluid surrounding a small area (few elements about 3-5 of them). Using previous idea results of velocity field developed for the fluid and volume distribution mark elements that accomplish two of three criteria.

The velocity profile of the Fig. 8 suggests, that most of the defects must be in the lower left corner, that result most match with the less homogeneous volume distribution to predict de area with defects in the solidified ingot (this can be performed because the part is thin enough to observe the flow pattern).

Fig. 9 shows the areas where there was less homogeneity in the velocity profile and distribution of fluid volume, at the same time the place with higher probability of turbulence. In the lower left corner and at the fluid inlet none uniformity in volume distribution appears but not in velocity profile, this type of results indicate that the material will show poor homogeneity during the filling that results in oxide salts accumulation and not desirable microstructures. When comparison is made in the corresponding ingot areas, porosity and traces of oxide layers are presented, as a result of turbulence flow, since the solidification is not considered in the model only is possible to match instantaneous results with the solidified history on the final part. Finally, the surface turbulence does not show defects accurately, but it shows a clear tendency to accumulate fluid on the left and lower on the right, which was the expected profile.

Conclusions

The flow simulation of material from one cavity to another, driven by gravity is only satisfactory when the calculation of free surfaces, propagation material inside the domain defined, velocity profile and Eulerian distribution volume are valid; in this case the previous variables are acceptable and make evident that the defects in solid as a result of turbulence flow match with the predicted turbulent zones, as a result ABAQUSTM was set like alternative tool of CFD software and capable of multiphysics analysis because all elastic and fluid equations defined by user are solve simultaneously.

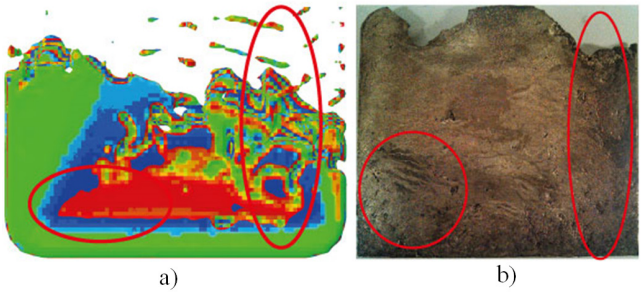

Figure 9 Comparison of a) EVF obtained (empty blue, red = full) b) solidified ingot (Photograph by Author).

GC algorithm with frictionless properties allow to control the surfaces interaction by the viscous properties defined on the fluid thus avoid "runoff" or bad geometric restriction by solid defined in the model. The material void may cause effects on the Eulerian material if they are under confinement, like at reduction of the funnel is needed to verify this behavior in other confined systems and other boundary conditions.

The combination of velocity profile and volume distribution allows to predict areas of high probability of turbulence that could be used to approximate zones with probability of defects, these predicted areas must be analyzed increment per increment because solidification effects are not considered, at the end of the simulation al areas are going to look homogeneous and many defects would pass unnoticed.

Acknowledgment

The authors would like to thank G. Álvarez, I. Cueva, L. Morales and E. Ramos, for technical assistance during the elaboration of this study.

This work was supported by Dirección General de Asuntos del Personal Académico, UNAM, under grants PE103816.

References

[1] B.Q. Li, *Solidification Processing of Materials in Magnetic Fields*, JOM (Journal of Materials), 50, no. 2, 1998

[2] 2. J. Campbell, *Castings*, Butterworth-Heinemann, 2nd edition, 2003.

[3] 3. M. Fleming, *Solidification Processing*, McGraw-Hill, New York, 1974.

[4] 4. J. Campbell, M. Holliday, B. Sirrel, *Benchmark Testing to solidification modeling the flow of Al castings*, JOM (Journal of Materials), 48, pp 20-23, 1996.

[5] H.M. Domanus., *Computer Simulation Using CAPS of an Aluminum Plate Casting, Modeling of Casting,* Welding and Advanced Solidification Processes VII, ed. M. Cross and J. Campbell Warrendale, PA: TMS, pp. 947-954, 1995.

[6] R.W. Lewis, K. Ravindran, and V. Tran, *Finite Element Simulation of the Benchmark Mold Filling Problem*, Welding and Advanced Solidification Processes VII, ed. M. Cross and J. Campbell Warrendale, PA: TMS, pp. 955-962, 1995.

[7] C. Rigaut, *Round Robin Bench Mark Exercise for Mold Filling and Solidification*, Welding and Advanced Solidification Processes VII, ed. M. Cross and J. Campbell Warrendale, PA: TMS, pp. 1007-1014, 1995.

[8] F. Sant and G. Backer, *Application of WRAFTS Fluid Flow Modeling Software to the Benchmark Test Casting*, Welding and Advanced Solidification Processes VII, ed. M. Cross and J. Campbell Warrendale, PA: TMS, pp. 983-990, 1995.

[9] Z.A. Xu and F. Mampaey, *Mold Filling and Solidification Simulation of the Benchmark Casting*, Welding and Advanced Solidification Processes VII, ed. M. Cross and J. Campbell Warrendale, PA: TMS, pp. 963-970, 1995.

[10] M.R. Barkhudarov and C.W. Hirt, *Casting Simulation: Mold Filling and Solidification—Benchmark Calculations Using FLOW-3D*, Welding and Advanced Solidification Processes VII, ed. M. Cross and J. Campbell Warrendale, PA: TMS, pp. 935-946, 1995.

[11] I. Ohnaka and J.D. Zhu, *Computer Simulation of Fluid Flow and Heat Transfer of the Benchmark Test by DFD-M/3DFLOW*, Welding and Advanced Solidification Processes VII, ed. M. Cross and J. Campbell Warrendale, PA: TMS, pp. 971-974, 1995.

[12] M.A. Layton et al., *Modeling of Mold Filling and Solidification*, Welding and Advanced Solidification Processes VII, ed. M. Cross and J. Campbell Warrendale, PA: TMS, pp. 975-982, 1995.

[13] D. M. Stefanescu (Volume Chair), *ASM Metals Handbook: Casting*, ASM International ® The Materials Information Company, Vol 15, 9th edition, 1992.

[14] R. Avila, *Numerical simulation of phase change problems (freezing-melting) with natural convection*, Ingeniería Mecánica Tecnología y Desarrollo, Vol. 1, No. 4, 2004.

[15] L. Fan, D. Peiran, S. Steve, *Coupled Fluid/Structure Interaction Simulation Using Abaqus CEL*, Dassault Systemes Simulia Corp, 2009.

[16] J. Tippmann, S. Prasad, P. Shah, *2-D Tank Sloshing Using the Coupled Eulerian-Lagrangian (CEL) Capability of ABAQUS ® / Explicit*, Dassault Systemes Simulia Corp, 2009.

[17] X. Latorre, P. Marijuan, J. Viñas, *Road Restraint Systems in ABAQUS ®*, IDIADA Automotive Technology SA, 2012.

[18] M. Ahmadzadeh, B. Saranjam, A. Hosseini-Fard, AR. Binesh, *Numerical simulation of sphere water entry problem using Eulerian-Lagrangian method*, Applied Mathematical Modelling, 38 (5-6): 1673-1684 (2014).

[19] R. Martínez, *Comportamiento Macro y Micro Mecánico de aleaciones con memoria de forma del tipo Cu-Al-Ni con manganeso y niobio bajo tensión uniaxial*, Tesis, Instituto de Investigación en Materiales, Universidad Nacional Autónoma de México, 2012.

[20] Simulia ABAQUS® 6.12, *Eulerian analysis of a collapsing water column*, ABAQUS® Benchmarks Manual, Section 1.7.1, 2012.

[21] Simulia ABAQUS® 6.12, *Specifying frictional behavior for contact mechanical property options*, Abaqus ®/CAE User's Manual Section 15.14.1, 2012.

[22] Simulia Abaqus Technology Brief, *Water Landing of Space Flight Re-entry Vehicles Using Abaqus / Explicit*, TB-11-SD-1, 2011

[23] A. Saucedo, *Simulación de la colada gravitacional mediante paquetería de elemento finito*, Tesis, Facultad de Ingeniería, Universidad Nacional Autónoma de México, 2013.