MOLD FILLING SIMULATION USING A CONTROL VOLUME FINITE ELEMENT METHOD

André Luíz de Souza Araújo
Mechanical Engineering Departament
Federal University of Ceara
P. O. Box 12.144
Campus do Pici
60.455-760, Fortaleza, CE, Brazil
e-mail: araujoals@yahoo.com

José Maurício Alves de Matos Gurgel
Departament of Mechanical Technology
Federal University of Paraiba
P. O. Box 5.115
58.051-970, João Pessoa, PB, Brazil
e-mail: gurgel@les.ufpb.br

Francisco Marcondes
Mechanical Engineering Departament
Federal University of Ceara
P. O. Box 12.144
Campus do Pici
60.455-760, Fortaleza, CE, Brazil
e-mail: marconde@dem.ufc.br

Abstract. *Mold filling injection is a widely used procedure for production of plastic parts. The realistic simulation of the filling process allows optimization of the mold design, significantly reducing manufacturing costs. The flow in mold injection is characterized by a continuously moving boundary formed between the injected fluid and that initially in the mold, whose location must be determined. The capture of the front can be achieved by Eulerian and Lagragian methods. The Volume of Fluid (VOF) is one of the most used Eulerian methods. The main task of the present work is to present the Control Volume Finite Element Method (CVFEM) for the solution of fluid flows in mold cavities, and to apply the VOF method to model the moving front. Results are presented to a radial open-mold.*

Keywords*:* Mold filling, VOF method, CVFEM**.**

1. Introduction

Mold filling injection and casting are manufacturing procedures widely used for production of plastic and metal parts. Realistic filling process simulations allow the engineer to optimize the mold design. However, the modeling of such problems presents several major difficulties due to several particular characteristics. One of these features is the existence of a continuously moving boundary between the injected fluid and the air that initially fills the mold cavity. The location of this boundary (free surface) is not known *a priori*, and its tracking can be modeled through several algorithms, which can be divided into two main categories according to the gridding, Jun and Spalding (1988), Maliska and Vasconcelos (1999), Araújo (2004): Lagrangian methods (grids adjust to the moving fronts) and Eulerian methods (grids are fixed).

This work presents the solution of a two dimensional mold filling process for a radial open-cavity. The governing equations are solved by the Control Volume Finite Element Method (CVFEM), and the injected fluid/air interface is modeled by using an Eulerian scheme based on the widely used VOF method, Hirt and Nichols (1979). In this approach, the location of the free surface is calculated by solving an additional differential equation to evaluate the fraction of the control volume filled with the injected fluid. The technique of Swaminathan and Voller (1994) is employed in order to reduce the numerical diffusion that characterizes the VOF method. The results are presented in terms of front position, pressure, and velocity along a radial cavity using structured and unstructured meshes using irregular rectangles.

2. Mathematical model

Considering the resident and injected fluid as incompressible and isothermal, and laminar flow, the governing equations representing the conservation of mass, and momentum equations for two-dimensional geometries, employing the Einstein summation, are given by

$$\frac{\partial}{\partial x_j}\left(\rho u_j\right) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(\rho u_i\right) + \frac{\partial}{\partial x_j}\left(\rho u_j u_i\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + s_{ui} \tag{2}$$

where *i* and *j* range from 1 to 2, ρ is the specific mass, μ is the viscosity, *u* and *v* are the Cartesian components of the velocity vector, *p* is the thermodynamic pressure, and s_{ui} is the source term of the momentum equation.

The flow is considered isothermal, incompressible and the injection fluid, Newtonian.

2.1. The VOF method

In the Volume of Fluid method (VOF), developed by Hirt and Nichols (1979), the fraction (*F*) of the computational cell occupied by the injection fluid is calculated through an additional differential equation,

$$\frac{\partial}{\partial t}\left(\rho_l F\right) + \nabla\cdot\left(\rho_l \vec{V} F\right) = 0 \tag{3}$$

where *t* is the time, ρ_l is the specific mass of the liquid injected into the mold cavity and $\vec{V}$ is the velocity vector.

A value of *F* equal to zero indicates an empty cell, between zero and one, partially filled, and equal to one, a filled cell. This approach has been applied in several situations with success. Usmani et al. (1992), and Codina et al. (1994) implemented this procedure in a finite element framework in order to simulate the casting of metal parts into complex shaped molds. Hétu et al. (1998) employed the VOF technique to study the injection of plastic in tri-dimensional molds. However, since the VOF equation, Eq. (1), is governed by convection alone, its discrete representation is propensity to the usual numerical problems associated with the handling of convection terms: spurious oscillations and the so-called numerical diffusion, Patankar (1980), Maliska (2004), which causes the smearing of the free surface. In order to reduce this effect and to preserve a sharp interface between the injection fluid and the air, Swaminathan and Voller (1994) proposed the following modification of Eq. (3),

$$\frac{\partial}{\partial t}\left(\rho_l F\right) + \nabla\cdot\left(\rho_l \vec{V} G\right) = 0 \tag{4}$$

where the parameter *G* is set to zero if $F < 1$. The main idea of this approach is that there is no flow at the cell boundaries while the cell is filling.

2.1.2. Solution of the governing equations for two fluids

According to Swaminathan and Voller (1994), the momentum equation for a combination of two fluids is given by

$$\frac{\partial}{\partial t}\left(\overline{\rho}_F \vec{V}\right) + \nabla\cdot\left(\overline{\rho}_G \vec{V}\vec{V}\right) - \nabla\cdot\left(\overline{\mu}_G \nabla\vec{V}\right) + \nabla p + \vec{g}\overline{\rho}_F = 0 \tag{5}$$

where ρ is the specific mass, $\vec{V}$ is the velocity vector, μ is the viscosity, *p* is the thermodynamic pressure and $\vec{g}$ is the acceleration due to gravity. The barred quantities are used to represent averaged properties, e.g.,

$$\overline{\rho}_F = F\rho_l + (1-F)\rho_g \tag{6}$$

$$\overline{\rho}_G = G\rho_l + (1-G)\rho_g \tag{7}$$

where ρ_l is the density of the liquid and ρ_g, the density of the gas (air that initially fills the mold cavity). For a compressible flow, the continuity equation reduces to,

$$\nabla\cdot\vec{V} = 0 \tag{8}$$

Equation (5) handles the large variations in density across the gas-liquid interface, thus the solution can be carried out in the entire (gas plus liquid) domain.

In this work the full Navier-Stokes equation is maintained. The momentum equation is not simplified to the usual form used in mold filling, where the convective terms are neglected, Hieber and Shen (1979), Kennedy (1995).

3. The Control Volume Finite Element Method (CVFEM)

The discretization of the governing equations, Eq. (5) and (8), and the *F*-equation, Eq. (4), was done employing the Control Volume Finite Element Method (CVFEM), developed by Raw (1984). A detailed description of this methodology can also be found in Raw and Schneider (1986), Souza (2000) and Araújo (2004).

The CVFEM combines the advantages of two classic numerical procedures, the Finite Element Method (FEM) and the Finite Volume Method (FVM): ability of handling solution domains of great complexity, employing unstructured grids (FE), scheme strictly conservative, discretization of the governing equations is based on balances of the properties over an elementary volume (FEM). Another important feature of this method is the simultaneous solution of the mass conservation and the Navier-Stokes equations. This procedure avoids the strong coupling between the pressure and velocity fields, typical of numerical schemes that employ segregated methods, Patankar (1980), Maliska (2004)

3.1. Discretization of solution domain

The solution domain is subdivided into much smaller regions, the finite elements. In this work only distorted quadrilaterals were used. Nodes are distributed at every element corner and all the physical variables of the problems (velocity components, pressure, temperature and so on) are stored at each node, which implies a collocated arrangement. Within each element (which is dealt with in isolation), a local and no orthogonal coordinate system *s*-*t* is defined. The coordinates *s* and *t* range from –1 to 1, and the nodes are labeled from 1 to 4 (Fig. 1).

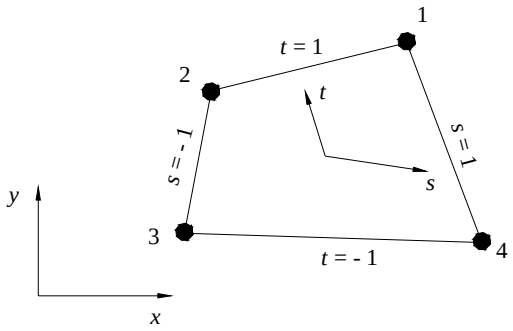


Figure 1. Finite element definition.

The element is called isoparametric, which means that all the problem variables, physical and geometric, are calculated within the element in terms of their nodal values using the same relation,

$$\Phi(s,\,t) = \sum_{i=1}^{4} N_i(s,\,t)\Phi_i \tag{9}$$

where Φ denotes any problem variable (u, v, x, y etc), Φ_i is the nodal value at node i, and the functions $N_i(s, t)$, called shape functions, are defined by

$$N_1(s,t) = \frac{1}{4}(1+s)(1+t) \tag{10}$$

$$N_2(s,t) = \frac{1}{4}(1-s)(1+t) \tag{11}$$

$$N_3(s,t) = \frac{1}{4}(1-s)(1-t) \tag{12}$$

$$N_4(s,t) = \frac{1}{4}(1+s)(1-t) \tag{13}$$

The x and y derivatives of a variable Φ are given by,

$$\left.\frac{\partial \Phi}{\partial x}\right|_{(s,\,t)} = \sum_{i=1}^{4} \left.\frac{\partial N_i}{\partial x}\right|_{(s,\,t)} \Phi_i \tag{14}$$

$$\left.\frac{\partial \Phi}{\partial y}\right|_{(s,\,t)} = \sum_{i=1}^{4} \left.\frac{\partial N_i}{\partial y}\right|_{(s,\,t)} \Phi_i \tag{15}$$

where,

$$\frac{\partial N_i}{\partial x} = \frac{1}{J}\left(\frac{\partial N_i}{\partial s}\frac{\partial y}{\partial t} - \frac{\partial N_i}{\partial t}\frac{\partial y}{\partial s}\right) \tag{16}$$

$$\frac{\partial N_i}{\partial y} = \frac{1}{J}\left(\frac{\partial N_i}{\partial t}\frac{\partial x}{\partial s} - \frac{\partial N_i}{\partial s}\frac{\partial x}{\partial t}\right) \tag{17}$$

and J is the Jacobian of the transformation between x-y and s-t.

The local derivatives of the shape functions can be determined from de Eq. (6) to (9).

A control-volume is established for every node in the solution domain. The lines $s = 0$ e $t = 0$ within each element surrounding a given node identify the control-volume edges (Fig. 2). The effective control-volume is formed, upon assembly, as the assemblage of contributions from all elements having that node in common.

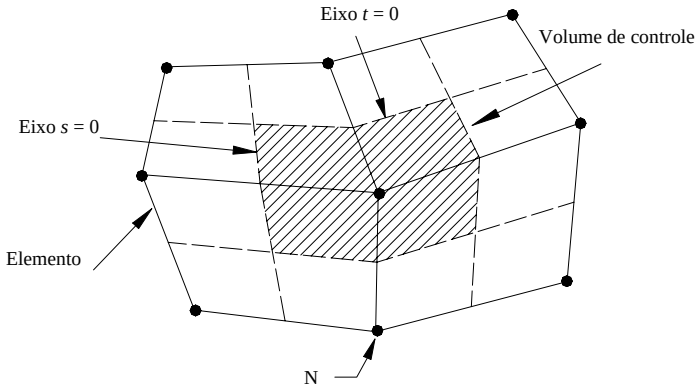


Figure 2. Control-volume definition.

The lines $s = 0$ and $t = 0$ also divide each element in four quadrants, called sub-control-volumes (SCV). In order to obtain the algebraic equations, the integrals of the convective and diffusive flows over the control-volume surfaces must be determined. These integrals are evaluated at the midpoints of each line segment that defines the control-volume surface. The midpoints are called integration points (ip) and the line segments, sub surfaces (SS), Fig. 3.

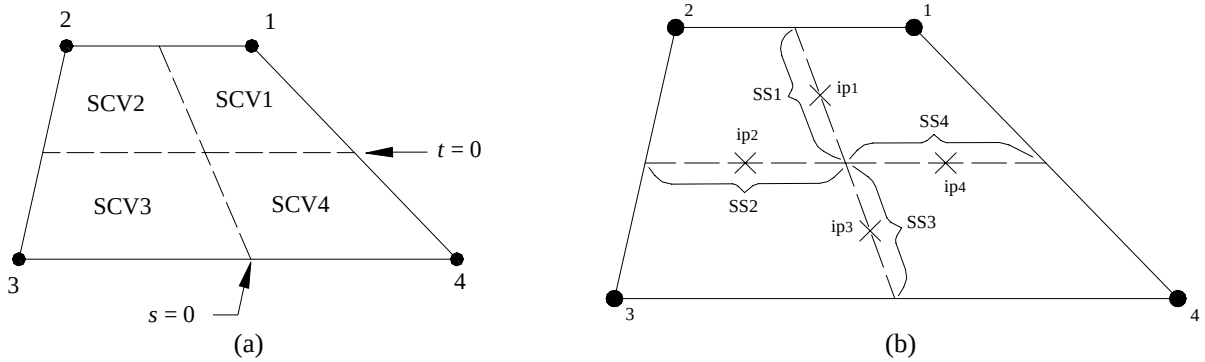


Figure 3. (a) Sub-control volume definition; (b) Integration points and sub surfaces definition.

3.2. Discretization of the governing equations

The basis of the CVFEM is that an algebraic statement representing conservation of each conserved quantity (momentum, mass, energy) is applied at every control-volume. The resulting algebraic equations will express a balance between the convective and diffusive flows of the variable in question and, if present, the transient and source terms.

In this approach, the equation of a control-volume is assembled from the four sub-control-volumes of the neighboring elements. The governing equations, Eq. (5) and (8), are integrated over the four sub-control-volumes of each element, and the following matrix relations are obtained,

$$\left[E^{uu}\right]\{U\}+\left[E^{uv}\right]\{V\}+\left[E^{up}\right]\{P\}\doteq\left\{R^{u}\right\} \tag{18}$$

$$\left[E^{vu}\right]\{U\}+\left[E^{vv}\right]\{V\}+\left[E^{vp}\right]\{P\}\doteq\left\{R^{v}\right\} \tag{19}$$

$$\left[E^{pu}\right]\{U\}+\left[E^{pv}\right]\{V\}+\left[E^{pp}\right]\{P\}\doteq\left\{R^{p}\right\} \tag{20}$$

where U is the x-velocity component, V is the y-velocity component, p is the thermodynamic pressure.

The procedure to determine the expressions for the matrix coefficients E^{uu}, E^{uv}, E^{up}, E^{vu}, E^{vv}, E^{vp}, E^{pu}, E^{pv}, E^{pp} and the vectors R^{u}, R^{v}, R^{p} can be found in Raw (1984), Raw and Schneider (1986), Souza (2000), Araújo (2004) and Araújo et. al (2004). In this matrix equation, each row represents the elemental SCV contribution to the corresponding nodal control-volume equation. Thus, Eq. (18) to (20) does not represent a single control-volume equation, but four contributions to four different control-volume equations. As mentioned earlier, the individual control-volume equation is completed upon assembly of all elemental contributions.

3.3. Solution procedure

The solution procedure for the determination of F is as follows,

1. Set the initial values of dependent variables (u, v, p). Set $F = 0$ for all volumes (empty cavity);
2. Solve the linear system, Eq. (15) to (17) and calculate the velocity e pressure fields;
3. Calculate F and G using the Eq. (4);
4. With F and G values, update the physical properties using Eq. (6) and (7);
5. Return to step (2) until the desired convergence is achieved, or return to step 2 for the next time step.

4. Results

The flow injection in a radial cavity is studied. This problem, which is depicted in Fig. 4, has analytical solution which can be used to check the numerical results.

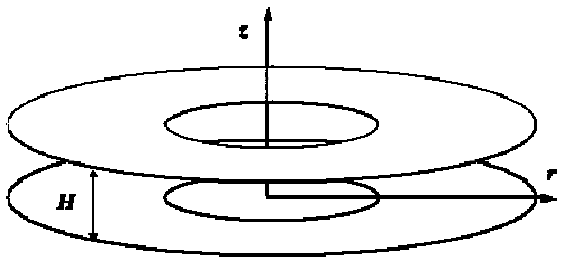


Figure 4. Geometry of the radial cavity.

If the distance H between the two discs is small, the flow can be considered as bi-dimensional, Maliska and Vasconcelos (1999). In this case, the analytical solution comes from the following equation, Maliska and Vasconcelos (1999), Araújo (2004),

$$\frac{d}{dr}\left(\mu r\frac{dp}{dr}\right)=0 \tag{21}$$

with $V_r = V_i$ at $r = R_0$ and $p = 0$ at $r = R(t)$, where V_r is the radial velocity, V_i is the injection fluid velocity (radial direction), R_0 is the inner radius and t is the time. The position $R(t)$ can be derived from the continuity equation,

$$R(t) = \sqrt{2V_i R_0 t + R_0^2} \tag{22}$$

The boundary conditions for the numerical solution were $V_r = V_i = 10$ m/s at $r = R_0 = 100$ mm and parabolic at $r = 200$ mm. Also, for the physical properties, it were used $\rho_g = 0.35$ kg/m^3, $\rho_l = 2{,}500$ kg/m^3, $\mu_g = 4.0 \times 10^{-5}$ kg/m/s and $\mu_l = 2.5 \times 10^{-3}$ kg/m/s, where the subscript l denotes the liquid and g the gas (air).

Two grids with 1,000 and 3,000 volumes were employed. Figure 5 shows the grid with 1,000 volumes. The time step was of 10^{-4} s.

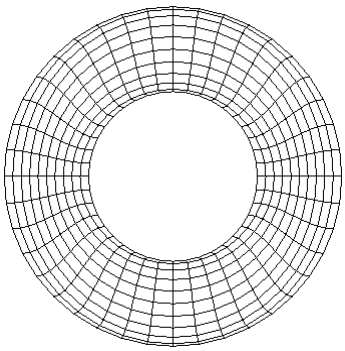
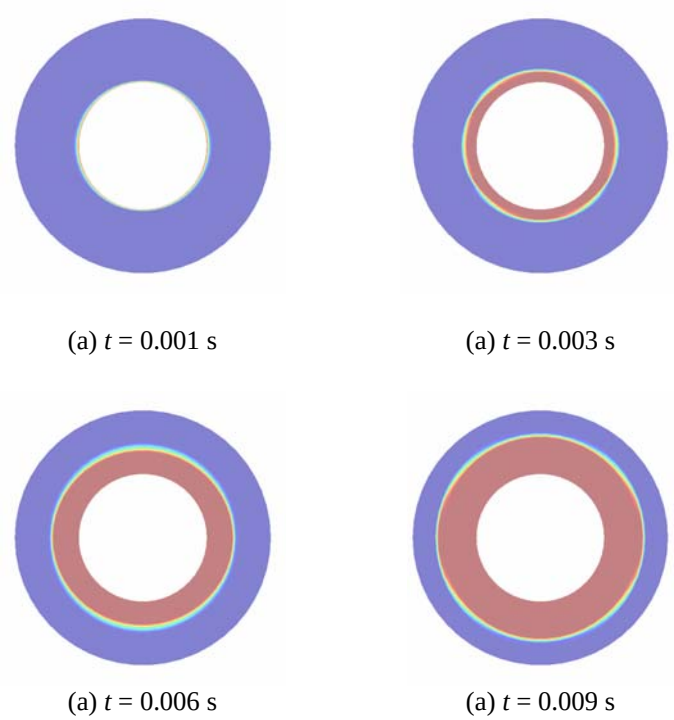


Figure 5. Grid with 1,000 volumes, radial cavity.

Figure 6 shows the position of the moving front during the filling process. From this figure it is possible to see that the obtained front is almost symmetric. Small differences can be occasioned to the post-processor code used to visualize the results.

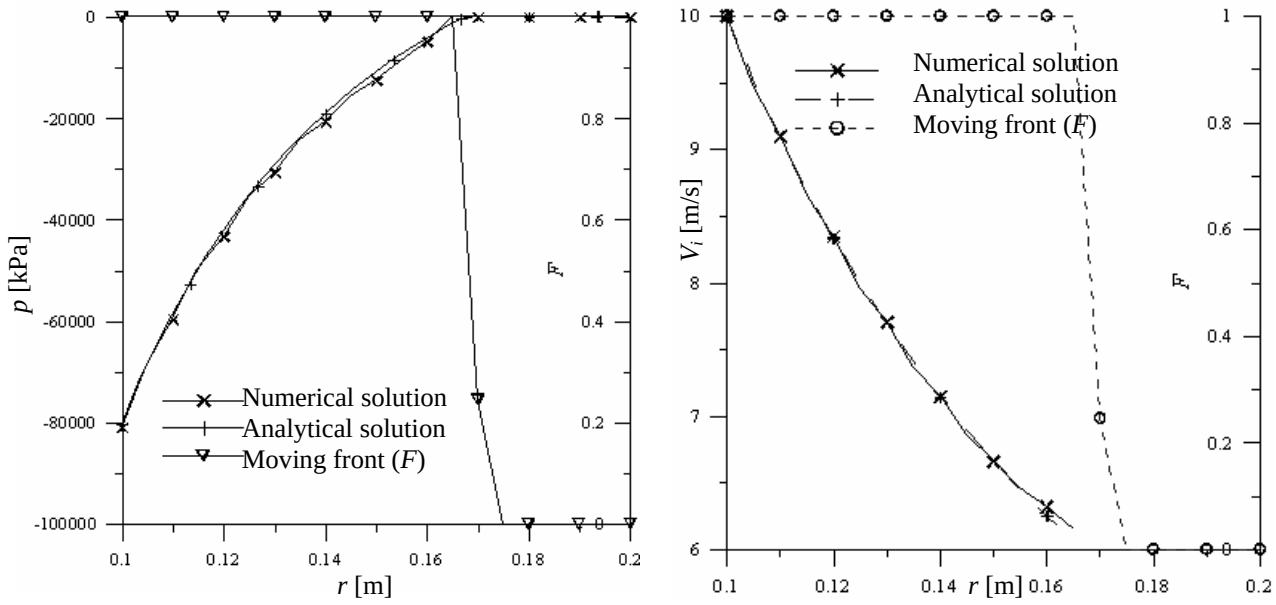


(a) $t = 0.001$ s (a) $t = 0.003$ s

(a) $t = 0.006$ s (a) $t = 0.009$ s

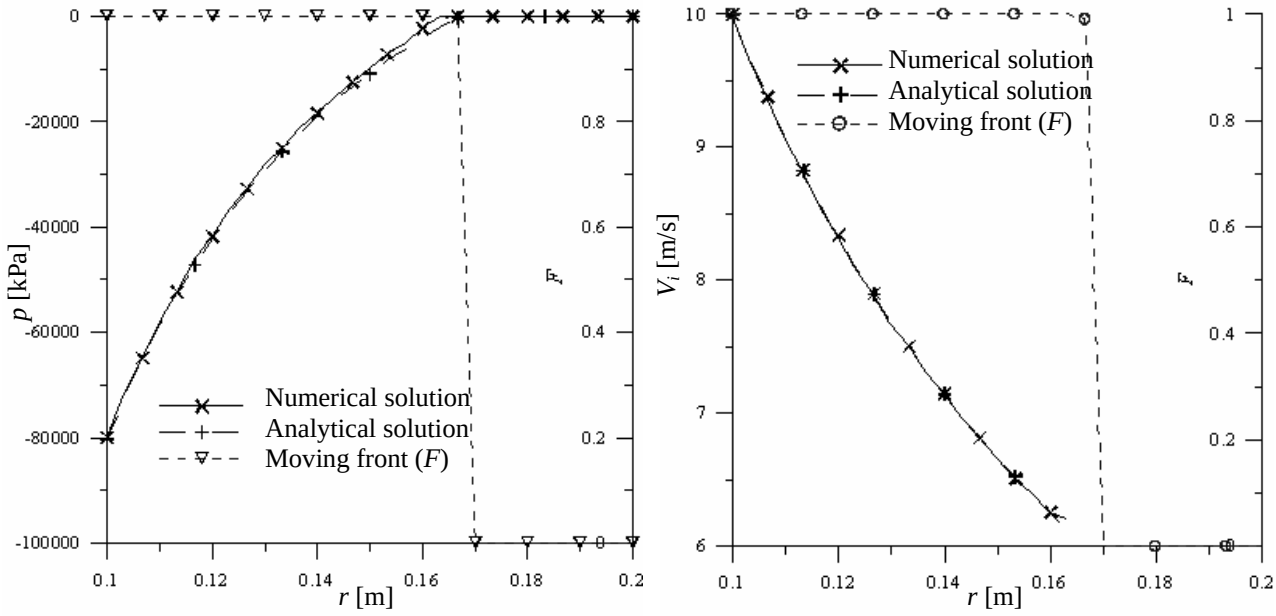
Figure 6. Prediction of the free surface location, radial cavity (grid with 1,000 volumes).

Figures 7 (grid with 1,000 volumes) and 8 (grid with 3,000 volumes) present the pressure and velocity fields at t = 0.009 s obtained numerically. These solutions are compared with the analytical solution given by Eqs. (21) and (22). From these figures it is possible to observe that a good match between analytical and numerical solutions was obtained. Even for a coarse grid (1,000 volumes) the pressure and front position were very close to that one given by the analytical equation. For instance, according to Eq. (22), for $R_0 = 0.1$ m, $V_i = 10$ m/s and $t = 0.009$ s, the moving front is located at $r = 0.167$ m. For the grid with 1,000 volumes, this value was of 0.165 m and for the grid with 3,000 volumes, 0.166 m.



(a) Pressure; (b) Radial velocity;

Figure 7. Comparison between numerical an analytical solution at t = 0.009 s, radial cavity (grid with 1,000 volumes).



(a) Pressure; (b) Radial velocity;

Figure 8. Comparison between numerical an analytical solution at t = 0.009 s, radial cavity (grid with 3,000 volumes).

5. Conclusions

The main goal of this work was to present the Control Volume Finite Element Method for the solution of the filling process. The moving boundary was modeled using the VOF method and the technique of Swaminathan and Voller (1994) to reduce numerical diffusion. The injection fluid was Newtonian, and the flow considered isothermal. The results were shown for a radial cavity.

The results (pressure, radial velocity and the location of the moving front) were compared to the available analytical solution. Two grids were employed, with 1,000 and 3,000 volumes, respectively, and in both cases, an excellent agreement were achieved between numerical and analytical solutions. Although the problem solved is very simple and has analytical solution, the authors believe that the employed methodology is capable of solving the complex problems

found in inject molding such as irregular geometries and non-linear physical properties. This type of methodology for injection molding is currently being investigated and the results will appear in future works.

6. References

Araújo, A. L. S., "Solução de Escoamentos de Fronteira Livre Usando Malhas Não-Estruturadas e Método dos Volumes Finitos Baseado em Elementos Finitos", thesis, Federal University of Paraíba, UFPB, Paraíba, Brazil, 2004.

Araújo, A., L. S., Gurgel, J.M., and Marcondes F., "Solução de Escoamentos Incompressíveis como Transferência de Calor Utilizando o Método dos Volumes Finitos Baseado em Elementos Finitos com Malhas Não-Estruturadas", X Brazilian Congress of Thermal Engineering and Sciences, ENCIT 2004, Rio de Janeiro, Brazil, 2004.

Codina, R., Schafer, U and Oñate, E., "Mould Filling Simulation Using Finite Elements", Int. J. Num. Meth. Heat Fluid Flow, Vol. 4, pp. 231-310, 1994.

Hétu, J. F., Gao, D. M., Garcia-Rejon, A. and Salloum, "3D Finite Elment Method for the Simulation of the Filling Stage in Injection Molding", Polymer Eng. Sci., Vol. 36 (2), pp. 223-236, 1998.

Hieber, C. A. and Shen, S. F., "A Finite Element/Finite Difference Simulation of the Injection Molding Filling Process", Journal of Non-Newtonian Fluid Mechanics, Vol. 7, pp. 1-32, 1979.

Hirt, C. W. and Nichols, B. D., "Volume of Fluid (VOF) Method for Dynamics of Free Boundaries", Journal of Computational Physics, Vol. 39, pp. 201-225, 1979.

Jun, L. and Spalding, D. B., "Numerical Simulations of Flows with Moving Interfaces", PhysicoChemical Hydrodynamics, Vol. 10 (5), pp. 625-637, 1988.

Kennedy, P. "Flow Analysis of Injection Molds", Hanser/Gardner Publications, 1995.

Maliska, C. R., 2004, "Transferência de Calor e Mecânica dos Fluidos Computacional", LTC, Rio de Janeiro, Brasil.

Maliska, C., R. and Vasconcelos, J. F. V., "An Unstructured Finite Volume Procedure for Simulating Flows with Moving Fronts", Comput. Methods. Appl. Mech. Engrg, Vol. 182 (3-4), 401-420, 1999.

Patankar, S. V., 1980, "Numerical Heat Transfer and Fluid Flow", Hemisphere/McGraw-Hill, New York, pp. 96-100.

Raw, M. J., "A New Control-Volume-Based Finite Element Procedure for the Numerical Solution of the Fluid Flow and Scalar Transport Equations", thesis, University of Waterloo, 1985.

Raw, M. J. e Schneider, G. E., "A Skewed Positive Influence Coefficient Upwinding Procedure for Control-Volume-Based Finite Element Convection-Diffusion Computation", *Num. Heat Transfer*, Vol. 9, 1-26, 1986.

Souza, J, A., "Implementação de um Método de Volumes Finitos com Sistema de Coordenadas Locais para a Solução Acoplada das Equações de Navier-Stokes", dissertação de mestrado, Universidade Federal de Santa Catarina, 2000.

Swaminathan, C. R. and Voller, "A time Implicit Filling Algorithm", Applied Mathematical Modeling Vol. 18, pp. 101-108, 1994.

Usmani, A. S., Cross, J. T. and Lewis, R. W., "A Finite Element Model for the Simulation of Mould Filling in Metal Casting and the Associated Heat Transfer", Int. J. Numer. Methods Eng., Vol. 35, pp. 797-806, 1992.

Vasconcelos, J. F. V., "Um Método de Volumes Finitos Usando Malhas Não-Estruturadas para o Estudo de Escoamentos com Frentes Livres", thesis, Federal University of santa Catarina, UFSC, Brazil, 1999.

7. Responsibility notice

The authors are solely responsible for the printed material included in this paper.