NUMERICAL SIMULATION OF THE RESIN TRANSPORT THROUGH FIBER REINFORCEMENT MEDIUM

Jeferson Avila Souza, jasouza@furg.br
Luiz Alberto Oliveira Rocha, laorocha@furg.br
Dep. de Física, Fundação Universidade Federal do Rio Grande, Av. Itália, km 8, Rio Grande-RS, 96201-900, Brazil

Sandro Campos Amico, amico@ufrgs.br
Dep. de Engenharia de Materiais, Universidade Federal do Rio Grande do Sul, Av. Bento Gonçalves, 9500, Bairro Agronomia, Porto Alegre-RS, 91501-970, Brazil

Abstract. *This paper describes the numerical simulation of the RTM (Resin Transfer Molding) process applied to the modeling of the resin transport through a fibrous reinforcement. The molding volume which is to be impregnated with the resin is considered as a porous medium and the Darcy equation is used to determine the resin transport velocity through the mold. A control volume finite element method is used for the determination of the pressure gradients inside the mold and the resin flow front advance is obtained using a FAN technique. The finite volume method was built to be used with a bi-dimensional unstructured grid, hence allowing the discretization of complex geometries. In the simulation presented here, resin physical properties, like viscosity and density, and the permeability of the media were kept constants.*

Keywords: *RTM, Porous Media, Finite Volume*

1. INTRODUCTION

RTM (Resin Transfer Molding) is a manufacturing process for composite materials with a number of important advantages, being a very versatile process for the efficient production of composites with complex forms and high structural performance. In the RTM process, a fibrous reinforcement is placed inside a closed mold and a polymeric resin is injected. The resulting product has uniform density with good surface finishing. The process is nowadays used in large scale by the automotive and aeronautic industries.

The RTM process was commercially introduced in Brazil only in the end of the 80 years, but in terms of scientific development it is still incipient in Brazilian academic institutions.

Many works on the mathematical modeling and numerical simulation of the RTM process may be found in the literature. The mathematical formulation includes, if all physical phenomena are considered, the momentum, energy and mass balance equations applied to a heterogeneous porous media. The fluid (resin), in many cases, must also be considered as non-Newtonian fluid.

The main goal of the numerical simulation is to determine, with accuracy, the flow front advance of the resin inside the mold. This information can then be used for the determination of the time needed to fill up the mold with the resin and the identification of, for instance, possible locations for void formations inside the mold. The numerical simulation may also be used for process optimization, where many simulations are performed in order to determine the best operating conditions, injection point location and mold geometry, avoiding this way, many experimental runs when designing a new mold.

The most used approach for the modeling of the resin transport inside a porous media is to consider that the fluid flow properties can be determined by the Darcy´s Law, which associates the velocity of the fluid with the pressure drop inside the porous medium. The pressure field is obtained by combining the mass conservation equation and the proposed Darcy's equation, resulting in a single differential equation for the pressure (see section 2). There are many methods used to solve the second order differential equation developed for the pressure (Eq. 5), but only a few simple cases, with specific fluid flow and geometry characteristics, have a closed (analytical) solution (Ballata *et al.* 1999; Modi *et al.*, 2003) and therefore a numerical technique is usually used to solve this equation.

There are many numerical approaches available for the pressure field determination, and the most frequently used formulations are based on: boundary elements, finite differences, finite elements or finite volumes. In the boundary elements formulation (Papatahnasiou, 2001), only the fluid flow front line is discretized. The pressure distribution along this front line is obtained via numerical integration over the boundaries of the resin flow. This information (pressure) is then used in the determination of the velocity field and the new fluid front line. The difficulty with this method is that for each time step, a new grid needs to be generated.

The most popular method used is that based on a finite element formulation (Bruschke and Adavani, 1990; Shojaei *et al.*, 2004), being well known in the literature as Finite Element/Control Volume Method - FE/CVM. In this method, the pressure field is obtained with a finite element formulation while control volumes, created around the grid nodes, are used for the calculation of the flow front advance. A surface integration along the control volumes limits is then used for the calculation of the flow-rate entering or leaving each control volume, and a fill factor is defined for the determination of the completely filled-up volumes. Consequently, the new flow front line may be defined. This method has also two important advantages: (i) a fixed grid is used in the computational domain discretization avoiding a new grid creation at each time step, and (ii) an unstructured grid can be used in the discretization of complex geometries.

In a similar way to the FE/CVM, where the pressure equation is solved with a finite element method, it is also possible to use a finite volume method to obtain the pressure field inside the computational domain (Jinlian *et al.*, 2004). One of the advantages of using control volume methods is that the fluxes of the properties (e.g. mass, momentum, etc.) through the control volume is already included in the basic formulation of the method, and no modifications in the code are necessary to account for the resin flow rates along the control volumes. Another advantage of using a control volume method is that, by definition, all the fluxes (including mass and species) along the control volumes must be conserved. New formulations of the finite volume methods, specially those known as Control Volume Finite Element - CVFE, also include the ability to deal, on its original formulation, with unstructured grids, allowing grids with triangular and rectangular (2D) or tetrahedral and hexadecimal (3D) elements, which are very important for the discretization of complex geometries.

Another type of method, called Volume of Fluid, VOF, "creates" a transport equation for the fill factor *f*. This new transport equation includes only the transient and the advective terms and is solved together with the other equations of the model (momentum, mass, energy, etc.). A discretization method (finite element, finite volume or finite difference) is then used to solve this system of partial differential equations resulting in the determination of pressure, velocity and fill factor fields inside the computational domain. This method is largely used for the computational simulation of the injection of thermoplastic materials into metallic molds, but can also be applied to the RTM process simulation. Examples of the application of this method are the unstructured finite volume solution proposed by Maliska and Vasconcellos, 1998 and the finite element solution applied to the RTM process used by Mohan *et al.*, 1999.

Finally, it should be mentioned that the pressure equation can also be solved with the finite differences method (Nielsen and Pitchumani, 2002). This method uses the Taylor series for the direct discretization of the differential equations of the mathematical formulation. Its main advantage relies on its simple discretization of the differential equations, therefore, when complex geometries must be discretized, this advantage becomes less effective and, depending on the complexity of the geometry, the effort to create the computational code may be comparable with that needed for more elaborated formulations, like finite element or finite volume methods.

In this work, a computational code is developed for the numerical simulation of the resin transport phenomena encountered in the RTM injection process. The pressure field is obtained with a control volume finite element method (Schneider and Raw, 1987), the velocity of the resin is calculated by the Darcy's Law equation and the determination of the flow front line advance inside the mold follows the methodology proposed by Frederick and Phelan (1997).

2. PROBLEM FORMULATION

The problem addresses the modeling of resin transport through a fibrous reinforcement media. The mold volume which is to be impregnated with the resin is considered as a porous medium and the Darcy law formulation is used to determine the resin transport through the mold.

Experimental observations performed by Darcy showed that the fluid velocity through a column of porous material is proportional to the pressure gradient established along the column (Bejan, 2004). The mathematical formulation for this phenomenon can be expressed as

$$\vec{V} = \frac{\bar{\bar{K}}}{\mu} \nabla p \qquad (1)$$

where $\vec{V}$ is the velocity vector (m/s), μ is the viscosity (Pa s) and $\bar{\bar{K}}$ is the permeability tensor (m²).

The continuity equation for an incompressible fluid takes the form of

$$\nabla \cdot \vec{V} = 0 \qquad (2)$$

Combining Eqs. (1) and (2)

$$\nabla \cdot \left(\frac{\bar{\bar{K}}}{\mu} \nabla p \right) = 0 \qquad (3)$$

For an isotropic medium and a Newtonian fluid with constant physical properties, Eq. (3) becomes

$$\nabla^2 p = 0 \qquad (4)$$

The boundary conditions to be used with Eq. (3), represented schematically in Fig. 1, are given by

(a) $P = P_0$ at the injection point;

(b) $\frac{\partial P}{\partial n} = 0$ at the mold walls (n is the direction normal to the wall); and

(c) $P = P_f$ at the fluid flow front line, where P_f is the front line specified pressure.

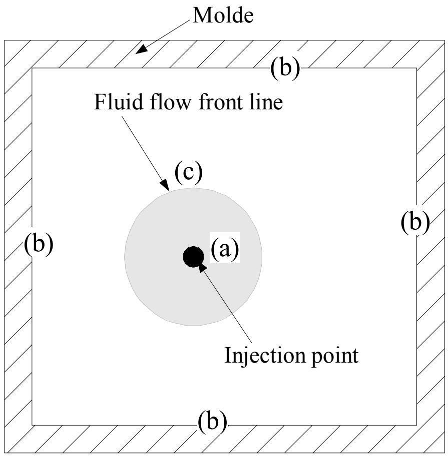


Figure 1 – Schematic representation of the injection problem

The solution of Eq. (3) inside the gray region of Fig. 1 provides the pressure field gradient between the injection point and the region not yet impregnated with the resin, therefore the main goal of the simulation is to determine the fluid front position as a function of the injection time. Since Eq. (3) does not include a transient term, the transient problem is solved by obtaining a steady state solution of Eq. (3) for each time step. This solution consists of dividing the computational domain into a number of finite volumes, which are initially considered empty (no resin). Next, from the known resin flow conditions of the previous time step (or boundary condition at the first time step), the filling rate of the volumes adjacent to the front line is calculated and the time for filling these adjacent volumes can be estimated. Then, these new filled volumes will compose a new flow front line for the resin inside the mold. A prescribed pressure boundary condition (P_f) is set to the volumes at the front line and Eq. (3) is again solved for the domain limited between the injection points (and walls) and the flow front line. The calculated pressure field is used to determine, with Eq. (1), the resin velocity field. Then, the mass flow-rate passing through the control volumes can be calculated. This procedure is repeated until the mold is totally full.

The numerical solution of Eq. (3) is obtained with a control volume finite element method (Souza e Maliska, 2000; Souza, 2000). The pressure field obtained by solving Eq. (3) is then used in Eq. (1) for the determination of the resin velocity inside de mold. Since the numerical solution of Eq. (3) is normally easily obtained, a significant part of the computational cost is spent on the determination of the advance of the flow front line.

The advancing of the flow front was obtained with a Flow Analysis Network (FAN) technique proposed by Frederick and Phelan (1997). First, the smallest time step needed to fill at least one volume is calculated by

$$\left(\Delta t\right)_{min} = min\left(\frac{\forall_i - \forall_i^f(t)}{\dot{\forall}(t)}\right) \tag{5}$$

where $\forall_i$ is the total volume of the finite volume *i*, $\forall_i^f(t)$ is the filled volume at time *t*, and $\dot{\forall}(t)$ is the volumetric flow-rate into volume *i*.

The $\left(\Delta t\right)_{min}$ is then used for the determination of the filling volume fraction and a filling factor *f* for the control volumes is defined as

$$f\left(t+\Delta t\right) = \frac{\forall_i^f(t) + \Delta t \cdot \dot{\forall}_i(t)}{\forall_i} \tag{6}$$

A point of the grid will be considered at the flow front when at least one of the sub-volumes of that element shows a filling factor equal to one. For example, in the grid of Fig. 2, the sub-volume hatched belongs to a fully filled control volume ($f = 1$), thus the points *j*, *k* and *l* should be considered at the flow front line and a pressure prescribed boundary condition ($P = P_f$) should be applied to them.

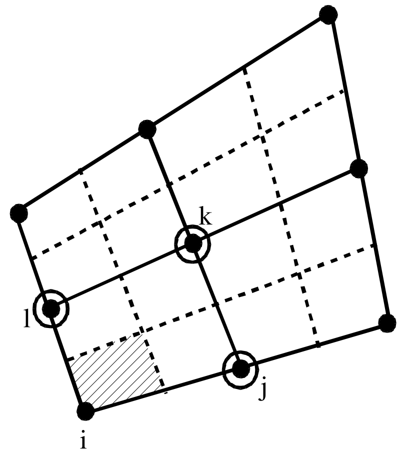


Figure 2 – Flow front line determination

Figure 3 shows a control volume created around node *1* which is formed with the contribution of four sub-control volumes (SVC), each one from a different element. The *s* and *t* variables represent the element local coordinates system. It can also be observed in Fig. 3 that the unstructured grids may be formed with irregular quadrilateral elements of different sizes, allowing the easy discretization of complex geometries.

The pressure field defined by Eq. (3), in the great majority of cases, does not have a closed (analytical) solution, thus a numerical procedure must be used for the solution of this equation. The methodology used in this work is very similar to that of well-known finite element/control volume-FE/CV methods. In this kind of methodology, the pressure field is obtained with a finite element solution of Eq. (3). The calculated pressure field is then used in Eq. (1) for the calculation of the velocity field and consequently the volumetric flow rate through the control volume surfaces. Therefore, in the present work, the solution of Eq. (3) is obtained with a finite volume method. The method used (Schneider and Raw, 1987; Souza and Malika, 2000) has already defined on this basic formulation both the elements and the control volumes needed to the present solution. The computational domain is discretized in elements, each one composed of four nodes (points) and the control volumes are created around the nodes with the contribution of four different elements. The quantity of the flow properties passing by each volume is accounted for by integrating the balance equations (continuity, momentum and energy) through the volume boundaries (surfaces).

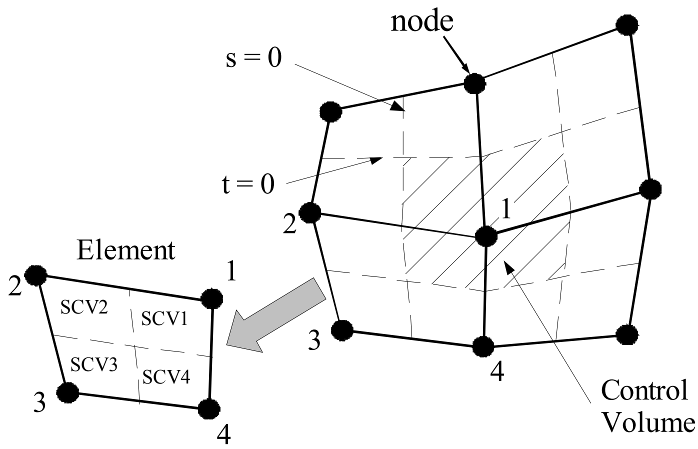


Figure 3 – Control volume creation

3. RESULTS

The proposed methodology and developed code was validated by the solution of two simple problems with known analytical solution. The problems used are: (i) rectilinear resin transport through a rectangular mold and (ii) radial injection from the center of a square preform.

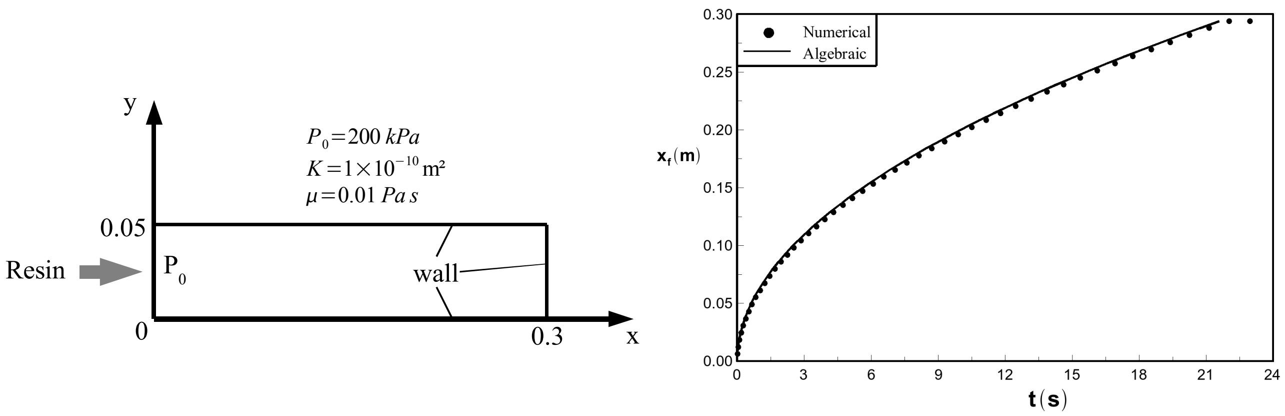
4.1. Rectilinear resin transport through a rectangular mold

Equation (5) was used to solve a simple porous media problem where a rectangular mold is boundary injected from the left side, see Fig. 4a. For constant injection pressure condition, the flow front position x_f at a time t can be calculated by (Jinlian, 2004)

$$x_f = \sqrt{\frac{2 K P_0 t}{\mu}} \tag{7}$$

where P_0 is the injection pressure [Pa] and t is the time of injection [s].

The solution was obtained with an 890-element grid. The predictions of the flow front position are in good quantitative agreement with the analytical solution, as shown in Fig. 4b.

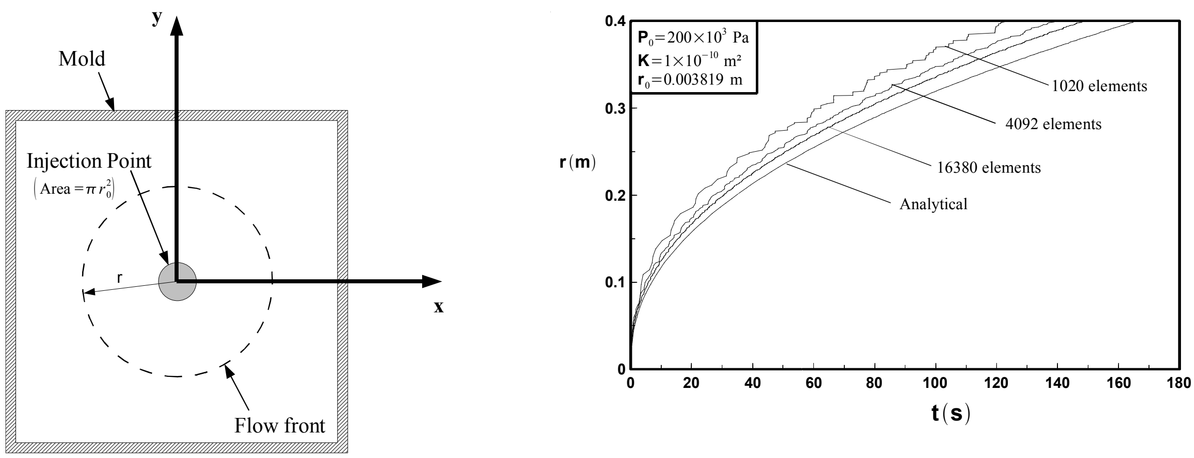


(a) (b)

Figure 4 – Boundary injection from one side of a rectangular mold: (a) problem sketch, and (b) analytical and numerical solution comparison

5.2. Point injection from the center of a square preform

The second problem type is described in Fig. 6a. In this problem, the resin injection is obtained by the application of a prescribed pressure P_0 at the center of the geometry. The resin will follow a radial flow from the center towards the mold walls.



(a) (b)

Figure 5 – Point injection from the center of a square preform (a) problem sketch, (b) analytical and numerical solution comparison

In this case, for a prescribed injection pressure P_0, the relationship between the flow front radial position r and the injection time t is given by (Rudd, 2005)

$$t = \frac{\mu}{2\,K\,P_{inj}}\left[r^2 \ln\left(\frac{r}{r_0}\right) - \frac{1}{2}\left(r^2 - r_0^2\right)\right] \tag{8}$$

where r_0 is the radius of the circular injection port.

The analytical and numerical comparison is shown in Fig. 5b. One can observe that the solutions are very similar at the first meters and that an accumulated error propagates with time, reaching a relative error of approximated 10% at r = 0.45m. This error was already reported in the lecture (Dai *et al.* , 2003; Joshi *et al.* 2000) and may be reduced with a grid refinement as showed in Fig. 5b.

6.3. Case study

The developed application was used in the simulation of the resin flow front advance in a rectangular mold with an internal injection point located near the left wall (Fig. 6a). A preset pressure P_0 was specified at the injection port and the simulation was carried out up to the complete filling of the mold with resin. The media and resin properties are also shown in Fig. 6a.

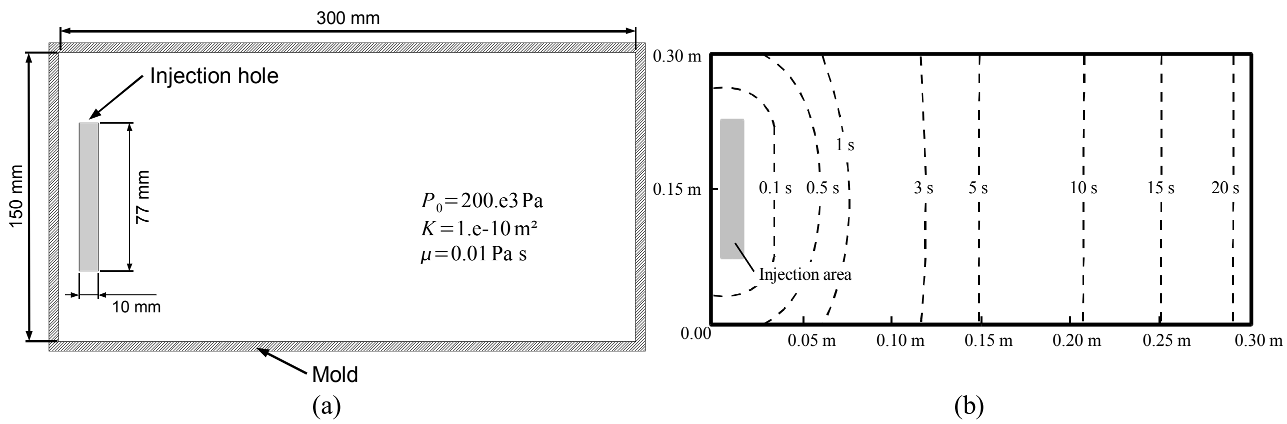


Figure 6 – Mold injection simulation (a) problem sketch, (b) flow front at different injection time

Figure 6b shows the flow front advance at different injection times during the simulation. It can be observed in this figure that the flow front advance is radial at the beginning of the impregnation process, but assumes a one-dimensional aspect at x approximated equal to 0.1m.

In the continuation of this research, an experimental RTM facility will be used to validate the developed methodology with actual experimental data and the influence of temperature on resin transport inside the mold will also be investigated.

7. CONCLUSIONS

The main goal of this work was to report the preliminary development of an application for the numerical simulation of the RTM process. This is a currently advanced technology world-wide and some commercial softwares are already available, although in Brazil just a few companies and researches are working with this kind of simulation, mostly using these softwares to solve a specific problem, and the country lacks researchers working on the basic analysis of the numerical methodology needed to simulate the RTM process.

In this context, a computational code has been developed for the simulation of the resin transport phenomenon inside a mold with fibrous reinforcement. The developed program uses a traditional approach where the pressure equation is solved numerically with a finite element solver, actually a control volume finite element solver, and the flow front advance of the resin is obtained with the methodology proposed by Frederick and Phelan (1997). Two problems with closed analytical solution were used for the validation of the developed code and a third problem, which will be lately compared with experimental data, was also simulated. The comparison between the analytical and numerical solutions showed good qualitative and quantitative agreement, showing the potential of the developed code to be used as a tool for the RTM process.

8. ACKNOWLEDGEMENTS

The authors acknowledge with gratitude the support of CAPES (PROCAD n° 030305) and FAPERGS – Fundação de Amparo à Pesquisa do Estado do Rio Grande do Sul.

9. REFERENCES

Bejan, A., 2004, "Convection Heat Transfer", John Wiley & Sons, Hoboken, New Jersey.

Ballata, W. O., Walsh, S. M., Advani, S., 1999, "Determination of the transverse permeability of a fiber preform", Journal of Reinforced Plastics and Composites, vol. 18, n. 16, pp. 1450-1464.

Bruschke, M. V., Advani, S. G, 1990, "A finite-element control volume approach to mold filling in anisotropic porous-media", Polymer Composites, vol. 11, n. 6, pp. 398-405.

Dai, F., Zhang, B., Du, S., 2003, "Analysis of Upper- and Lower-limits of Fill Time in Resin Transfer Mold Filling Simulation", Journal of Composite Materials, vol. 38, n. 13, pp. 1115-1136.

Frederick, R., Phelan, A., 1997, "Simulation of the Injection Process in Resin Transfer Molding", Polymer Composites, vol. 18, n. 4, pp. 460-476.

Jinlian, H., Yi, L., Xueming, S., 2004, "Study of Void Formation in Multi-layer Woven Fabric", Composites - Part A: Applied science and manufacturing, vol. 35, pp. 595-603.

Joshi, S. C., Lam, Y. C., Liu, X. -L., 2000, "Mass Conservation in Numerical Simulation of Resin Flow", Composites: Part A, vol. 31, pp. 1061-1068.

Maliska, C. R., and Vasconcellos, J. F. V. de, 2000, “An unstructured finite volume procedure for simulating flows with moving fronts”, Computer Methods in Applied Mechanics and Engineering, vol. 182, pp. 401-420.

Modi, D., Simacek, P., Advani, S., 2003, "Influence of injection gate definition on the flow-front approximation in numerical simulations of mold-filling processes", International Journal for Numerical Methods in Fluids, vol. 42, n. 11, pp. 1137-1248.

Mohan, R. V., Ngo, N. D., Tamma, K. K., 1999, "On a Pure Finite-Element-Based Methodology for Resin Transfer Mold Filling Simulations", Polymer Engineering and Science, vol. 39, n. 1, pp. 26-43.

Nielsen, D. R., Pitchumani, R., 2002, "Closed-loop flow control in resin transfer molding using real-time numerical process simulations", Composites Science and Technology, vol. 62, pp. 283-298.

Papathanasiou, T. A., 2001, "The hydraulic permeability of periodic arrays of cylinders of varying size", Journal of porous media, vol. 4, n. 4, pp. 323-336.

Rudd, C. D., 2005, "Liquid Moulding Technologies: Resin Transfer Moulding, Structural Reaction Injection Moulding, and Related Processing Techniques", SAE International.

Schneider, G. E., Raw, M. J., 1987, "Control volume finite element procedure for heat transfer and fluid flow using collocated variables: 1. Computational procedure", Numerical Heat Transfer, vol. 11, n. 4, pp. 363-390.

Shojaei, A., Ghaffarian, S. R., Karimian, S. M. H., 2004, "Three-dimensional process cycle simulation of composite parts manufactured by resin transfer moldin", Composite Structures, vol. 65, pp. 381-390.

Souza, J. A., 2000, Implementação de um Método de Volumes Finitos com Sistema de Coordenadas Locais para a Solução Acoplada das Equações de Navier-Stokes, Dissertação de Mestrado, UFSC, Florianópolis, SC, Brasil.

Souza J. A., Maliska C. R., 2000, Analysis of a volume based finite element methodology in view of the interpolation function employed and coupling characteristics, Proceedings of the 8th Brazilian Congress of Engineering and Thermal Sciences-ENCIT, Porto Alegre, Brazil.

10. RESPONSIBILITY NOTICE

The author are the only responsible for the printed material included in this paper.