SOME FIRST ORDER UPWIND TECHNIQUES FOR UPDATING THE FLUID DOMAIN IN FIXED MESH SIMULATIONS OF LCM MANUFACTURING PROCESSES

F. SÁNCHEZ[1], R. TORRES[2], J.A. GARCÍA[2], F. CHINESTA[3]

[1] *Universidad Cardenal Herrera CEU. Escuela Univ. Superior de Enseñanzas Técnicas*
C/ San Bartolomé 55, E-46115 Alfara del Patriarca, Valencia (Spain). Email: sanchezf@uch.ceu.es

[2] *Universidad Politécnica de Valencia. Dpto Ing. Mecánica y Materiales*
Camino Vera s/n, 46022, Valencia (Spain). Email: rtorres@dimm.upv.es, jugarcia@mcm.upv.es

[3] *Laboratoire de Mecanique des Systemes et des Procedes, UMR CNRS - ENSAM – ESEM,*
151 Boulevard de l'Hopital, 75013 Paris (France). Email: francisco.chinesta@paris.ensam.fr

Keywords: Fixed mesh techniques; Advection equation; Upwind techniques; Composite forming processes; RTM; Liquid injection moulding

ABSTRACT

Liquid Composites Molding (LCM) processes simulation involves a treatment of the flow front movement during the mold filling stage. In this work we propose a fixed mesh strategy proceeding in the whole domain, where a null pressure is prescribed in the empty region and the variational formulation associated with the Darcy's equation is enforced in the domain occupied by the fluid. In order to capture the flow front location, and in consequence, to predict the fluid domain evolution, a new variable defining the fluid presence is introduced. This variable takes a unit value in the fluid domain, it is zero in the empty region, and its evolution is governed by a scalar linear advection equation. This work focuses on the accuracy evaluation of different techniques for solving the linear advection equation governing the evolution of the fluid presence function.

INTRODUCTION

Liquid Composites Moulding (LCM) processes and, particularly, resin transfer molding (RTM) are being increasingly used in the manufacture of fiber-reinforced composite materials. Many of the final qualities of the piece are determined during the injection. Therefore, numerical simulation can help us to optimize the design and manufacture. A good composite part is obtained when the reinforcement is fully impregnated by the resin.

LCM processes simulation involves a treatment of the flow front movement during the mold filling stage. Two family of techniques exist to treat moving boundaries: the first one concerns the moving mesh strategies whose main drawback is the necessity of frequent remeshing to avoid large mesh deformations, whereas the second one, widely used, are the fixed mesh strategies, where the flow front is captured from the solution computed in a mesh related to the whole domain [1-2]. Some of the strategies in the second family proceed by solving the flow governing equations only in the part of the domain occupied by the fluid, whereas other ones solve the governing equations in the whole domain, requiring, obviously, the definition of a pseudo-behavior to apply in the empty zones.

In this work we propose a fixed mesh strategy proceeding in the whole domain, where a null pressure is imposed in the empty region, and the variational formulation associated with the Darcy's equation is enforced in the domain occupied by the fluid. In order to capture the flow front location, and in consequence, to predict the fluid domain evolution, a new variable defining the resin presence is introduced. This variable takes a unit value in the fluid domain, it is zero in the empty region, and its evolution is governed by a scalar linear advection equation. In order to simulate the filling process we applied a fully explicit technique, which proceeds as follows: the flow kinematics is computed in a given fluid domain, which will be updated from the kinematics just computed. The main advantage of this explicit strategy is that it allows us to decouple the elliptic character of the equations of motion and the hyperbolic character of the advection equation governing the fluid domain evolution, as well as its simplicity and its significant reduction of the degrees of freedom. This work focuses on the accuracy evaluation of different techniques for solving the linear advection equation governing the evolution of the resin presence function.

FLOW GOVERNING EQUATIONS

The resin impregnation is usually modeled as a flow in porous media. The model equations describing this flow, are given by:

- Darcy's law, which states that the flow velocity is proportional to the pressure gradient according to

$$\underline{v} = -\frac{\underline{\underline{K}}}{\mu}\nabla P\,, \tag{1}$$

where $\underline{v}$ is the velocity, $\underline{\underline{K}}$ is the preform permeability tensor, μ is the fluid viscosity and P is the pressure.

- The incompressibility of the fluid, is expressed by:

$$Div\ \underline{v} = 0\,, \tag{2}$$

- The evolution of the resin volume fraction I (resin presence function), is governed by the following linear advection equation:

$$\frac{dI}{dt} = \frac{\partial I}{\partial t} + \underline{v}\cdot\nabla I = 0\,, \tag{3}$$

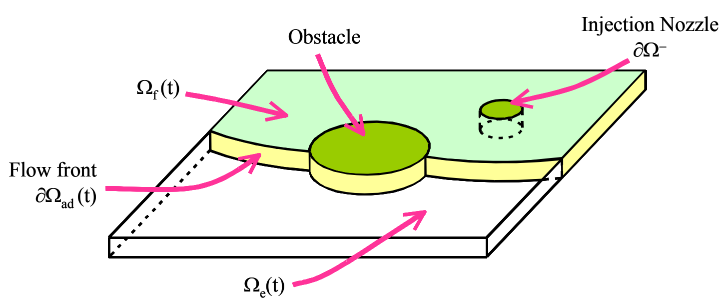


Fig. 1. Two-dimensional model.

The flow problem is defined in the mold volume Ω,

$$\Omega = \Omega_f(t) \bigcup \Omega_e(t)$$

where the fluid at time t occupies the volume $\Omega_f(t)$ and $\Omega_e(t)$ defines de empty part of the mold. The boundaries of $\Omega_f(t)$ and $\Omega_e(t)$ are denoted by $\partial\Omega_f(t)$ and $\partial\Omega_e(t)$ respectively. The filled and empty domains have a common boundary, which corresponds to the fluid flow front $\partial\Omega_{ad}(t)$ (see Fig. 1).

The location of the fluid into Ω is defined by the characteristic function I associated with the fluid and given by

$$I(\underline{x},t) = \begin{cases} 0 & \underline{x} \in \Omega_e(t) \\ 1 & \underline{x} \in \Omega_f(t) \end{cases}$$

If Eq. (1) is introduced into Eq. (2), the pressure at each point in the fluid domain can be obtained from the solution of

$$Div\left(\frac{\underline{\underline{K}}}{\mu}\nabla P\right) = 0\,. \tag{4}$$

Assuming that the permeability and viscosity are constants and that the permeability is orthotropic, then, the permeability tensor can be written in a diagonal form in the principal directions and Eq. (4) becomes in the two dimensional case

$$\left(k_{xx}\frac{\partial^2 p}{\partial x^2} + k_{yy}\frac{\partial^2 p}{\partial y^2}\right) = 0\,, \tag{5}$$

where k_{xx} and k_{yy} are the preform permeability in the principal directions.

The boundary conditions are given by:

- The pressure gradient in the normal direction to the mold walls is zero, that is, the fluid cannot leave the mold cavity trough the mold walls.
- The pressure or the flow rate is prescribed on the inflow boundary (injection nozzle) $\partial\Omega^-$:
  $p(\underline{x} \in \partial\Omega^-) = P_i \quad or \quad \underline{v}(\underline{x} \in \partial\Omega^-) = \underline{v}_i$
  where $\partial\Omega^- = \{\underline{x} / \underline{v}(\underline{x}) \cdot \underline{n} < 0\}$ and $\underline{n}(\underline{x})$ is the unit outwards vector, defined on the boundary at point $\underline{x}$.
- Zero pressure on the flow front
  $p(\underline{x} \in \partial\Omega_{ad}(t)) = 0$

And if we assume that at time $t=0$, the mold is empty, the initial condition for the function I results:

$$I(\underline{x}, t=0) = \begin{cases} 0 & if\ \underline{x} \in \Omega \\ 1 & if\ \underline{x} \in \partial\Omega^- \end{cases}$$

NUMERICAL MODELLING

The numerical resolution of the governing equations will be performed by means of a conforming finite element Galerkin technique, whereas that for the fluid domain updating a volume of fluid (VOF) technique [3] will be used. The resolution scheme is based in solving until the complete filling of the mold, that is, while $\Omega_f(t) < \Omega$, the next three steps:

1. Obtain the pressure field using a finite element discretisation of the following variational formulation

$$\int_{\Omega_f(t)} \left(\nabla p^* \cdot \underline{\underline{K}}\nabla p\right) d\Omega = 0 \tag{6}$$

The discretisation is extended to the whole domain imposing a null pressure at the nodes where any neighbor element is full filled.

2. Compute the velocity field from Darcy´s law, Eq. (1)

3. Update the element volume fraction *I*, integrating the corresponding transport equation given by Eq. (3)

RESIN DOMAIN UPDATING STRATEGIES

The advection equation, which governs the fluid presence function, *I,* is given by Eq. (3), as indicated previously. An accurate Eulerian discretisation technique for treating Eq. (3) consists in applying the Lesaint-Raviart technique (first-order discontinuous finite element method) as described in [4]. In order to apply this discretisation technique we write the conservation form of Eq.(3),

$$\int_{\Omega^e} \left(\frac{\partial I}{\partial t} + Div\left(I \underline{v}\right) - I \; Div \; \underline{v} \right) d\Omega = 0 \tag{7}$$

where Ω^e represents an element of a finite element mesh. From Eq. (7), using the divergence theorem and taking into account the fluid incompressibility, results

$$\int_{\Omega^e} \frac{\partial I}{\partial t} d\Omega + \int_{\partial\Omega^{e+}} I \underline{v} \cdot \underline{n} dS + \int_{\partial\Omega^{e-}} I \underline{v} \cdot \underline{n} dS = 0 \tag{8}$$

where $\partial\Omega^{e+}$ and $\partial\Omega^{e-}$denote the outflow and inflow boundaries of the element Ω respectively (see Fig. 2).

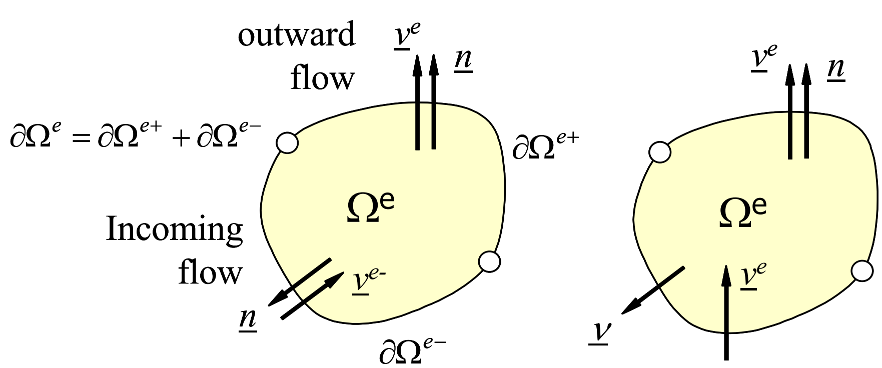


Fig. 2. Inflow and outflow boundaries

The main difficulty related to Eq. (8) is that the function *I* and the velocity $\underline{v}$ are constant into each element, and then, they are not defined on the element boundaries along which are applied the boundary integrals. The discontinuous finite element method assumes that on the outflow boundary the function *I* is the existing value inside the element Ω^e, i.e. $I\left(\underline{x} \in \partial\Omega^{e^+}\right) = I^e$, and that on the inflow boundary the function *I* is given by its value in the upstream element, i.e. $I\left(\underline{x} \in \partial\Omega^{e^-}\right) = I^{e^-}$.

Thus, Eq. (8) can be written in the equivalent form

$$\frac{\partial I^e}{\partial t}\left|\Omega^e\right| = -I^e \int_{\partial\Omega^{e+}} \underline{v} \cdot \underline{n} dS - I^{e-} \int_{\partial\Omega^{e-}} \underline{v} \cdot \underline{n} dS \tag{9}$$

where $|\Omega^e|$ denotes de volume of Ω^e.

If we consider a first order explicit approximation of the time derivative, we can write Eq. (9) as

$$I^e(t+\Delta t) = I^e - I^e \frac{q^+\Delta t}{|\Omega^e|} + I^{e^-} \frac{q^-\Delta t}{|\Omega^e|}, \tag{10}$$

where q^+ and $q^-$are the outflow and inflow rates, and all terms in the second member are evaluated at time t (explicit strategy).

From now on, we define the inflow and outflow fluid volumes as $\Omega^- = q^-\Delta t$ and $\Omega^+ = q^+\Delta t$. Thus, we can write finally

$$I^e(t+\Delta t) = I^e - I^e \delta^e \frac{\Omega^+}{|\Omega^e|} + I^{e^-} \delta^{e^-} \frac{\Omega^-}{|\Omega^e|} \tag{11}$$

where we have assumed that $\Omega^- \neq 0$ if, and only if, $I^{e-}=1$ and $\Omega^+ \neq 0$ if, and only if, $I^{e+}=1$, by using the parameter δ defined by:

$$\begin{cases} \delta^e = 1 & if \quad I^e = 1 \\ \delta^e = 0 & if \quad I^e < 1 \end{cases}$$

As in the pressure approximation a P_1-C^0 interpolation is considered, the velocity is constant into each element, and then, it becomes discontinuous across the element boundaries, which introduces additional difficulties in the fluxed computation.

The outflow volume term Ω^+ in Eq. (11) is calculated taking into account the fluid incompressibility (moreover, the velocity being constant into each element, its divergence becomes null)

$$\int_{\partial\Omega^{e+}} \underline{v} \cdot \underline{n} dS = -\int_{\partial\Omega^{e-}} \underline{v} \cdot \underline{n} dS$$

so it can be defined by:

$$\Omega^+ = q^+\Delta t = \left(\int_{\partial\Omega^{e-}} \underline{v}^e \underline{n} dS \right) \Delta t \tag{12}$$

where $\underline{v}^e$ denotes de velocity vector in the element Ω^e.

The inflow volume term, Ω^- in Eq. (11) is calculated from

$$\Omega^- = q^-\Delta t = \left(\int_{\partial\Omega^{e-}} \left(\beta \underline{v}^{e-} + (1-\beta) \underline{v}^e \right) \cdot \underline{n} \ dS \right) \Delta t \tag{13}$$

where depending on the values of the parameter β, the incoming flow rate varies. We analyze the following three situations:

$$\beta = 0 \quad \Rightarrow \Omega^- = q^- \Delta t = \left(\int_{\partial\Omega^{e-}} \underline{v}^e \cdot \underline{n} \; dS \right) \Delta t$$

$$\beta = 0.5 \quad \Rightarrow \Omega^- = q^- \Delta t = \left(\int_{\partial\Omega^{e-}} \left(0.5 \underline{v}^e + 0.5 \underline{v}^{e-} \right) \cdot \underline{n} \; dS \right) \Delta t \qquad (14)$$

$$\beta = 1 \quad \Rightarrow \Omega^- = q^- \Delta t = \left(\int_{\partial\Omega^{e-}} \underline{v}^{e-} \cdot \underline{n} \; dS \right) \Delta t$$

where $\underline{v}^{e-}$ denotes de velocity in the upstream neighbor element. Fig. 2 represents the situations of b=1 on the left and b= 0 on the right.

NUMERICAL TESTS

In order to analyze numerically the different fluid domain updating strategies, a rectangular mould has been considered (see Fig. 3). The injection velocity is 0.01 m/s and the preform permeability and the resin viscosity are 10^{-9} m^2 and 0.1 Pa.s, respectively.

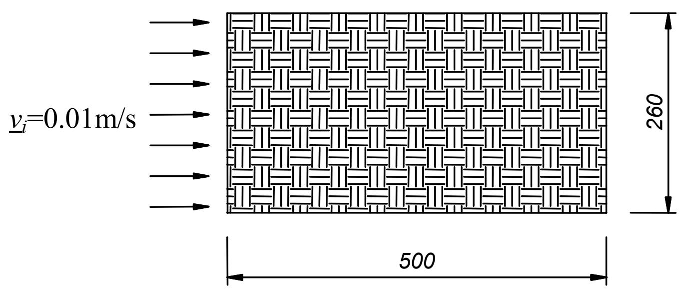


Fig. 3. Mould used in the numerical simulation.

Knowing the exact injected volume at time *t,* we can evaluate the accuracy on the numerical fluid domain updating. The error has been defined from the following expression:

$$Error(\%) = \left| \frac{AV(t) - IV(t)}{IV(t)} \right| \cdot 100$$

where *AV(t)* denotes the volume of fluid existing into the mould at time *t* (computed from the volume fraction solution at time *t*) and *IV(t)* denotes the exact injected volume at time *t* (computed directly from the inflow boundary condition). Fig. 4 shows the volume fraction distribution *I* for the three values of the parameter β at time *t*=30s. In that figure, the horizontal bars represent the exact and predicted (according to the unidirectional regime) flow front positions.

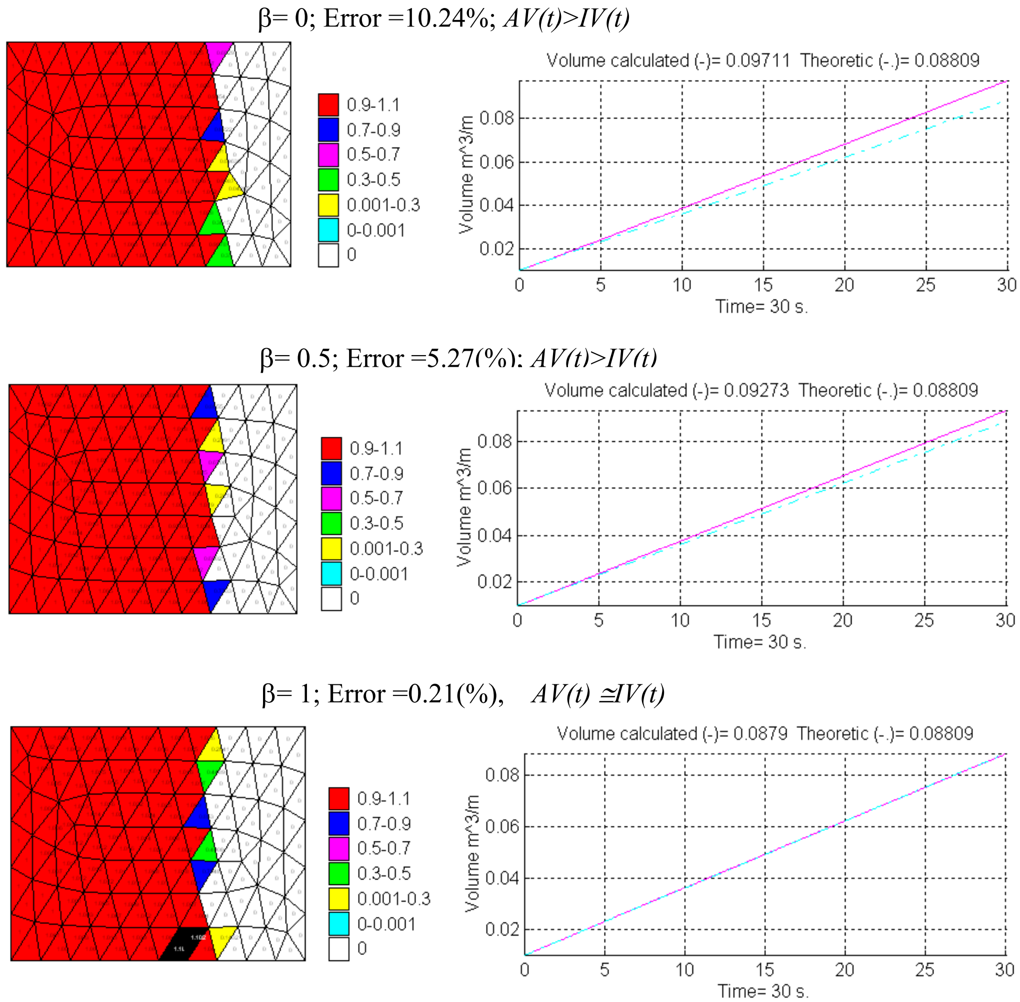


Fig. 4. Volume fraction distribution at time *t* =30s. with β=0, β=0.5, and β=1.

As a null pressure is imposed in the empty domain, some times, high numerical velocities can appear in the partially filled elements (see Fig. 5), with the associated overfilling effects when one uses a value of β lower than 1. It can be noticed that the use of the upstream element velocity $\underline{v}^{e-}$ to define the inflow flow rate (β=1), yields the best results in this particular unidirectional case.

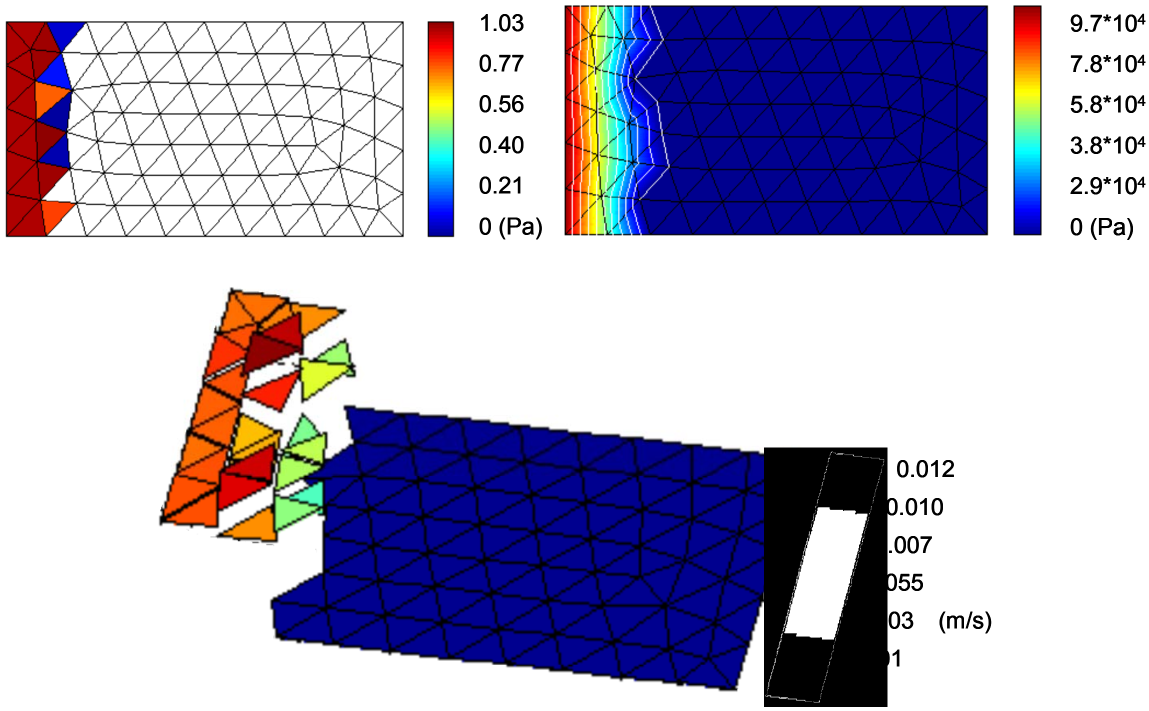


Fig. 5. Volume of fluid, pressure and velocity distributions at time t=2 s.

As expected, Fig. 6 proves that for a given time, in this case t=30 s, the solution accuracy increases as the number of elements increases, using a regular mesh. Moreover, the best results are obtained by using β=1 in Eq. (14).

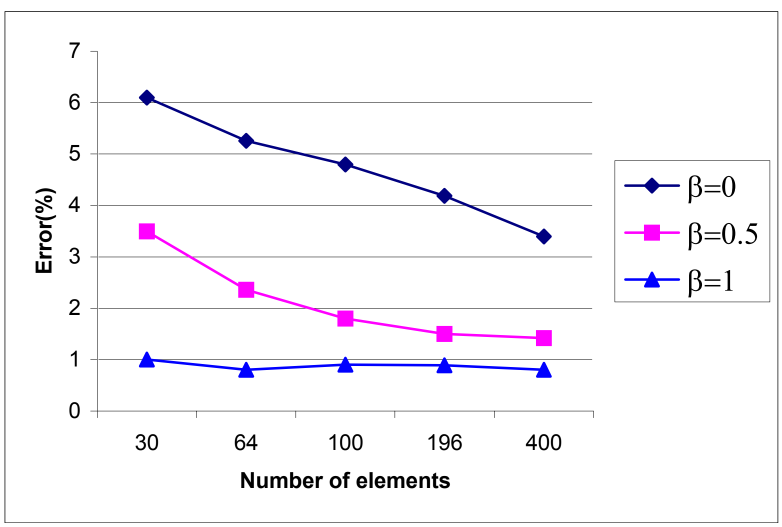


Fig. 6. Convergence analysis for a unidirectional flow

Another case is considered by including an obstacle inside the mould in order to introduce high velocity variations (in norm and in direction), as depicted in Fig. 7. In this case, as illustrated in Fig. 8, the best results are obtained for β= 0.5.

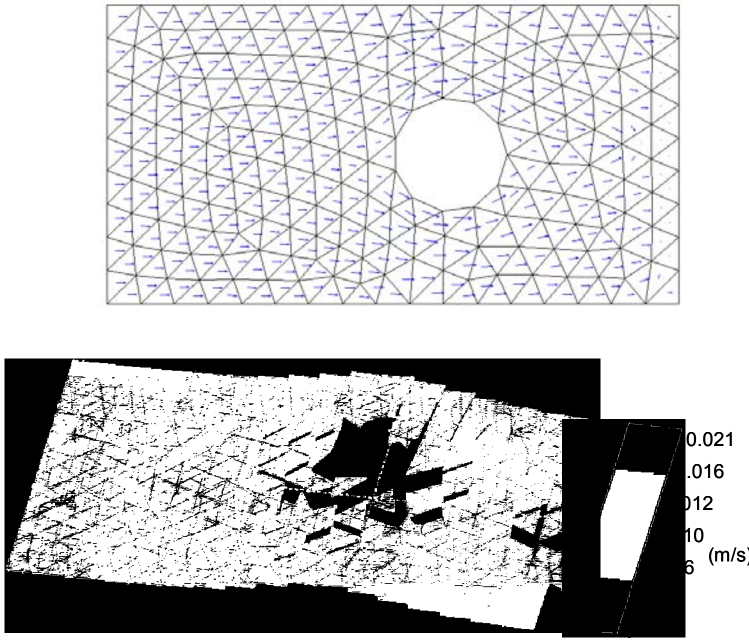
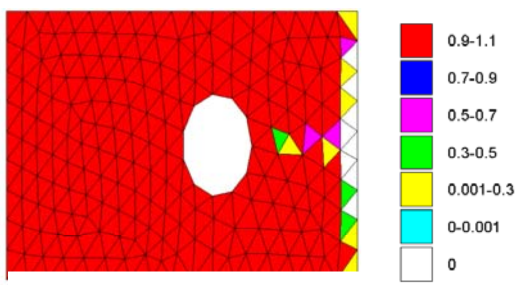
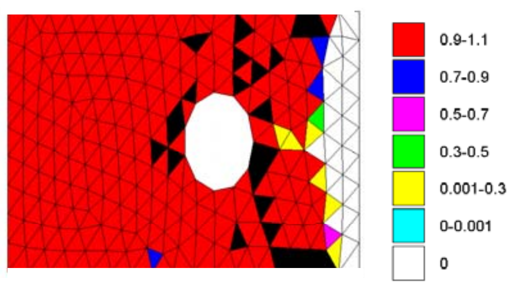


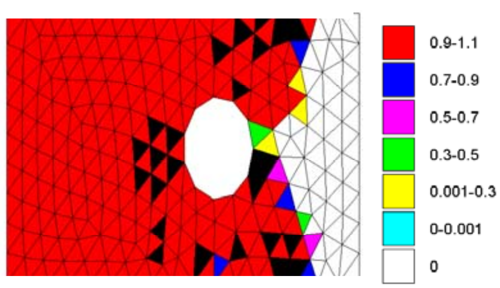
Fig. 7. Mould containing an obstacle: velocity prediction at time t=40 s. withβ=0.



β=0; Error =6.5%, *AV(t)>IV(t)*



β=0.5; Error =2.7%, *AV(t)>IV(t)*



β=1; Error =6.3%, *AV(t)<IV(t)*

Fig. 8. Mould containing an obstacle: volume fraction distribution at time *t*=40 s. (*β*= 0, *β*= 0.5 and *β*= 1)

Finally, if we consider a more complex situation as depicted in Fig. 9, the best results are then obtained in the case of β= 0.

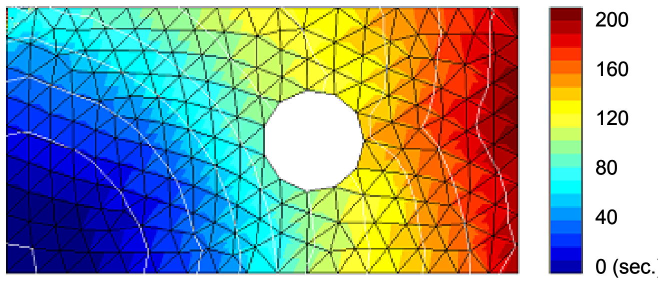


Fig. 9. Flow pattern for a filling time of 200 s. and β=0

CONCLUSIONS

Some numerical schemes for the resolution of the advection equation governing the evolution of the resin presence function have been tested. It has been shown that depending on the considered problem: geometry, injection conditions, ... the optimal parameters change. Thus, at present, we cannot conclude about the optimal technique, which require more depth convergence analysis.

REFERENCES

1. Advani S.G.,*Flow and rheology in polymer composite manufacturing,* (Elsevier, 1994)

2. J.A. García, Ll. Gascón, F. Chinesta, *A Fixed Mesh Numerical Method for Modelling the Flow in Liquid Composites Moulding Processes Using a Volume of Fluid Technique*, Computer Methods in Applied Mechanics and Engineering, 192/7-8 (2003) 877-893.

3. C. Hirt, B. Nichols, *Volume of fluid method for the dynamics of free boundaries*, J. Comput. Phys, 39, (1981) 201-225.

4. F. Chinesta*, Modélisation numérique en mise en forme des polymères et céramiques: différents problèmes de transport*, HDR, LMT ENS de Cachan (2000).