HAL open science

A generic mixed FE-SPH method to address hydro-mechanical coupling in liquid composite moulding processes

Sébastien Comas-Cardona, Paul Groenenboom, Christophe Binetruy, Patricia Krawczak

To cite this version:

Sébastien Comas-Cardona, Paul Groenenboom, Christophe Binetruy, Patricia Krawczak. A generic mixed FE-SPH method to address hydro-mechanical coupling in liquid composite moulding processes. Composites Part A: Applied Science and Manufacturing, 2005, 36 (7), pp.1004-1010. 10.1016/j.compositesa.2004.11.009 . hal-01004835

HAL Id: hal-01004835

https://hal.science/hal-01004835

Submitted on 24 Feb 2024

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

A generic mixed FE-SPH method to address hydro-mechanical coupling in liquid composite moulding processes

S. Comas-Cardona[a], P. Groenenboom[b], C. Binetruy[a,*], P. Krawczak[a]

[a]Ecole des Mines de Douai, Technology of Polymers and Composites Department, 941 rue C. Bourseul, BP 838, 59508 Douai, France
[b]Engineering Systems International-BV, Gebouw Kortland, Nieuwe Tiendweg 11a, 2922 EN Krimpen aan den IJssel, The Netherlands

Resin Transfer Moulding (RTM) can be a very attractive single-step process for sandwich structure manufacturing. During an RTM injection, large deformation of the foam core material is present due to the pressure field developing while saturating the fibre-reinforced skins. Such effects induce poor dimensional accuracy of skins and core thickness. Moreover, the foam core crushing increases filling time and injected resin amount. In order to model this hydro-mechanical coupling, the use of a mixed Finite Element and Smoothed Particle Hydrodynamics (FE-SPH) method is presented in this study. This method combines Lagrangian particles and finite elements, modelling, respectively, the resin and the sandwich core. Computed pressure responses during the filling stage are compared to symmetric sandwich manufacturing test results. The potential of this original numerical method is also discussed.

Keywords: A. Layered structures; C. Computational modelling; C. Numerical analysis; E. Resin transfer moulding (RTM)

1. Introduction

Resin Transfer Moulding (RTM) can be a very attractive single-step process for sandwich structures manufacturing. The two fibre-reinforced skins and the core are laid up in a mould cavity. Both skins are saturated and bonded to the core either using a pressure bucket to perform an injection at constant pressure or using a piston to perform an injection at constant flow rate. There is no need for an extra adhesive bonding step compared to other sandwich manufacturing processes. During sandwich manufacturing using RTM, the pressure field developing while saturating the fibre-reinforced skins can lead to large shifting [1,2] or compression [2,3] of the core as found in experiments. When such injections are performed at constant flow rate, it has been shown that measuring pressure at the resin inlet can be an appropriate sensor for the detection of hydro-mechanical coupling during sandwich manufacturing [3]. Moreover, during resin injection stage, fibre reinforcements can also be submitted to compression [1,4]. Such effects are damaging to the part because they induce poor dimensional accuracy of both skin and core thickness. Also, filling time and injected resin amount can be increased due to the core crushing.

Previous works focused on addressing those manufacturing issues by modelling the mechanical behaviour of fibre reinforcement under compression [5,6] and the hydro-mechanical coupling involved in composite manufacturing processes using finite element methods (FEM) [2,7], mixed control volume/finite difference methods (FDM) [8] or a one-dimensional Lagrangian formulation [9]. Numerical simulation of fluid flow is most frequently performed using FDM or FEM within an Eulerian reference frame. When the geometry of the flow domain changes, due to a free surface or interfaces with moving structures, or for mixing of multiple fluids, special solutions are required within these numerical schemes. There also exist many numerical simulation tools for flow through porous media, but it is often not possible to combine them with flow in non-porous regions.

In this paper, it will be shown how the Smoothed Particle Hydrodynamics (SPH) method provides an attractive alternative solution to simulate flow-induced foam core

* Corresponding author. Tel.: +33 3 2771 2175; fax: +33 3 2771 2981.
E-mail address: binetruy@ensm-douai.fr (C. Binetruy).

compression during symmetric sandwich manufacturing based on Binetruy and Advani's former work [3]. Foam core shifting will not be taken into consideration. The SPH method has previously been used to simulate RTM filling [10] and gave fairly good qualitative results in terms of resin flow front profiles. In order to model this coupling, a mixed Finite Element and Smoothed Particle Hydrodynamics (FE-SPH) method has been chosen. An SPH package is available within the commercial simulation tool PAM-CRASH (ESI, France) and has been further developed to solve for viscous flow in porous media. The following section presents the fundamentals involved with flow within porous media and a brief description of the mixed FE-SPH method. A comparison with another simulation tool for isotropic flow during RTM filling will be presented. Then, pressure responses computed from the filling simulations will be compared to symmetric sandwich manufacturing test results where hydro-mechanical coupling is present. Finally, the potential of this alternative and generic numerical method will be discussed.

2. Fundamentals

2.1. Fluid flow dynamics within a porous medium

For transport of a liquid within porous media, the flow is dominated by the viscous drag the liquid experiences. Therefore, these effects need to be included whenever the liquid is inside a porous region. It is possible to derive the conservation of mass and momentum equations for flow in porous media using the volume averaging method on the conservation equations for a two-phase medium in which one of the materials is fixed in space [11–13]. Considering that the volume fraction available for the fluid remains fixed and is called porosity, constant fluid viscosity, Newtonian fluid and laminar flow, the general forms of the equations to be solved (conservation of mass and momentum) are, respectively:

$$\frac{\partial \eta}{\partial t} + \nabla\cdot(\eta\vec{u}) = 0 \tag{1}$$

$$\frac{\partial \eta\vec{u}}{\partial t} + \nabla\cdot\eta\vec{u}\vec{u} = -\phi\nabla p + \phi\mu\nabla(\nabla\cdot\vec{u}) + \eta\vec{g} - \phi\vec{f}_{\mathrm{d}} \tag{2}$$

where ϕ is the fluid volume fraction or porosity, $\vec{u}$ the fluid velocity, μ the viscosity, p the pressure, $\vec{g}$ the gravitational acceleration, $\vec{f}_{\mathrm{d}}$ the drag force and η the apparent density defined as:

$$\eta = \frac{m_{\mathrm{f}}}{V_{\mathrm{t}}} = \phi\rho \tag{3}$$

where m_{f} is the fluid mass within a control volume V_{t}, and ρ the fluid density. Even though the flow of resin during composite manufacturing may be considered as incompressible, the governing equations (Eqs. (1) and (2)) are written for a compressible fluid. The reason for this will be given in the following section.

According to Tucker et al. [11], for composite moulding processes the viscous term in Eq. (2) is negligible. These authors also mentioned that the drag force might be considered as linearly dependent on the fluid velocity when inertia is negligible

$$\vec{f}_{\mathrm{d}} = \phi\mu K^{-1}\cdot\vec{u} \tag{4}$$

where K is the permeability. Taking into account the previous assumptions and combining Eqs. (1) and (2) leads to:

$$\eta\left\{\frac{\partial\vec{u}}{\partial t} + \vec{u}(\nabla\cdot\vec{u})\right\} = -\phi\nabla p + \eta\vec{g} - \phi^2\mu K^{-1}\cdot\vec{u} \tag{5}$$

In this study, in order to simulate the flow for all regions of the computational domain, Eq. (5) provides the basis of the solution with the viscous drag term only being used within the porous domain. As a limiting case, within the porous region (i.e. where the flow is dominated by the viscous drag and the Reynolds number based on the pore size is much lower than 1) the LHS of Eq. (5) may be neglected and hence Darcy's equation may be derived

$$\vec{v} = -\frac{K}{\mu}(\nabla p - \rho\vec{g}) \tag{6}$$

where $\vec{v} = \phi\vec{u}$ is the transport velocity.

2.2. SPH formulation

The SPH is a Lagrangian and mesh less method which was originally developed to solve astrophysical problems [14,15] and has later been applied also to the flow of compressible fluids [16–20]. In SPH, the fluid is modelled by N particles of mass m_i. Each particle i will interact with all neighbouring particles j that belong to the sphere of influence of the particle i (Fig. 1). The sphere of influence is defined by:

- a kernel function W in order to weigh interactions according to the distance between the particles;
- a smoothing length h to limit interactions in space.

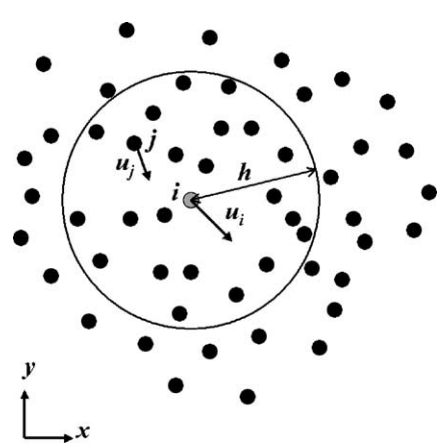

Fig. 1. Sphere of influence of a particle i.

Several kernel functions are available in the literature and in the used software. For this study, the cubic spline kernel has been chosen [21,22]. Note that the kernel functions have properties similar to the Dirac function. Another specificity of the SPH method is the particle size. It plays a similar role as the grid or mesh size in FDM or FEM: a smaller particle size implies a better spatial resolution of the flow, but more CPU. The smoothing length is defined to be proportional to the particle size.

Within the SPH method, particles represent a fixed amount of fluid. If the fluid enters a porous region, the mass has to be distributed over a larger volume in space. Since the fluid particles should be able to model the progress of the flow front, the proper choice for the SPH formulation in porous media is to let the particles move with the microscopic fluid velocity. Therefore, the advancement of the front is a direct consequence of the motion of the particles representing the resin, both inside and outside the porous region. This is one advantage of using the Lagrangian SPH description.

It is straightforward to derive the SPH formulation corresponding to Eqs. (1) and (5) [14–19,23] which in this case involves the apparent density and the microscopic velocity:

$$\frac{\mathrm{d}\eta_i}{\mathrm{d}t} = \sum_j m_j(\vec{u}_i - \vec{u}_j)\cdot\nabla_i W_{ij} \tag{7}$$

$$\frac{\mathrm{d}\vec{u}_i}{\mathrm{d}t} = -\phi_i \sum_j m_j \left. \frac{p_j}{\eta_j^2} + \frac{p_i}{\eta_i^2} + \Pi_{ij} \right) \nabla_i W_{ij} + \vec{g} - \phi_i^2 \mu K_i^{-1} \frac{\vec{u}_i}{\eta_i} \tag{8}$$

where W_{ij} is the SPH smoothing kernel, m_j the mass of the particle j. In order to stabilize the solution scheme, the standard Monaghan–Gingold artificial viscosity contribution Π_{ij} has to be added to the momentum equation [23]. It has been shown that when numerical shock phenomena are no longer dominating the solution, this artificial viscosity may be used to mimic the physical viscosity, but it may be difficult to identify the proper parameters [17]. The possibility to include non-Newtonian viscosity in the fluid flow is currently being investigated. For the current study, only the most relevant contribution of viscosity to Eq. (8), the viscous drag term, is included. Numerical solution of Eqs. (7) and (8) allows to model the flow in both porous and non-porous regions of a domain. When the particle is in the porous region, the RHS of Eq. (8) becomes very large and the equation becomes equivalent to Darcy's law. However, when the fluid particle is in a non-porous region, the viscous drag term is dropped. Orthotropic effects of the porous medium are easily incorporated by considering the permeability K to be a tensor in Eq. (8).

Usually, the flow of resin during moulding is considered as that of an incompressible fluid. In that case, Eq. (1) reduces to $\nabla\cdot\vec{u} = 0$ and the pressure should be obtained from a Poisson type of equation. Such a system of equations can be solved efficiently by an implicit method which allows for a relatively large time step. In the present study, however, the solution time step needs to be small. The first reason is that the motion of the flow front requires an accurate solution in time and space when the resin passes the border of the porous region. A second reason is that the weak coupling of the flow with the foam, as discussed below, requires an explicit solution with a sufficiently small time step. Hence, the original Eqs. (7) and (8) (corresponding to Eqs. (1) and (2)) will be solved in combination with an artificial equation of state (Murnaghan model) for nearly incompressible liquid [10,16–20]:

$$p = p_0 + B\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right] \tag{9}$$

with p_0 a reference pressure, ρ_0 the reference density, B the (artificial) bulk modulus and γ the exponent. The parameters B and γ of Eq. (9) are adjusted to keep density variations lower than a few percent. SPH solutions for incompressible flow have also been developed, but such solutions have not been pursued for this study.

2.3. Interaction of SPH particles with finite elements

The interaction of the particles with the finite elements, representing, respectively, the resin and the foam in this study, is modelled by the sliding interface algorithms available within PAM-CRASH [21]. Sliding interfaces model the interaction between structures and parts of structures that are not permanently connected by standard finite element connectivity conditions. Most of these sliding interfaces are based on the well-known penalty formulation, where geometrical interpenetrations between contacting faces are penalized by counteracting forces that are in essence proportional to the penetration depth. The contact algorithm will automatically detect when a particle (slave) penetrates any segments (master) of the outer surface of the finite element of the foam. The contact thickness indicates the distance away from a contact face where physical contact is established. In this case the contact thickness should be representative of the particle size. The use of such interaction between the SPH and finite elements has been validated for a range of applications such as sloshing [24,25], heart valve opening [26] and impact of aeronautical structures on water [20].

2.4. Numerical schemes

Due to the motion of a free surface and of the interface with the deformable structural parts (e.g. the foam core during sandwich manufacturing) the computational time step will be required to be small and an explicit solution scheme in combination with the nearly-incompressible fluid model will provide a suitable solution. Pressure,

gravity and viscous drag (through the viscosity to permeability ratio) contribute to the momentum equation (Eq. (8)). For low permeability, the drag force may become so high that the direct explicit solution becomes unstable. Therefore, the drag force contribution to the SPH momentum is integrated in an implicit manner. Furthermore, a special solution has been implemented for particles very close to the interface between the porous and non-porous regions in order to avoid numerical oscillations.

3. Comparison between the predictions of FE-SPH method and PAM-RTM

Before using the mixed FE-SPH method to solve for hydro-mechanical coupling during sandwich manufacturing, comparisons were made on flow front positions and pressure distributions with the commercial simulation tool PAM-RTM (ESI, France). The mould filling part of this software provides the solution for transient flow based on Darcy's law using linear non-conforming finite elements [27] and incorporates tracking of the flow front during injection. Following is an example of mould filling simulation with isotropic permeability. A Newtonian fluid is injected in a complex 2D mould containing 50% of fibre reinforcement (Fig. 2). The permeability K of the fibre reinforcement is isotropic and constant in time ($K = 10^{-10}$ m^2). The fluid viscosity is 0.001 Pa s and the fluid inflow velocity is 0.05 m s^{-1}. Fig. 2 shows the results of the simulations after 38 s (a and b) and 54 s (c and d) of injection. The comparison of the flow front positions using the SPH method (a and c) to the PAM-RTM ones (b and d) gives a good match. Minor differences in the filling sequence can be explained by the numerically enhanced compressibility used in the SPH simulations. Increasing the compressibility in SPH provides better agreement but requires higher computational effort. In terms of injection pressure responses, the maximal injection pressure obtained at the end of the injection is 0.07 and 0.08 MPa from SPH and PAM-RTM simulations, respectively. It is concluded that there is a quite reasonable agreement between the SPH method and PAM-RTM for porous media flow.

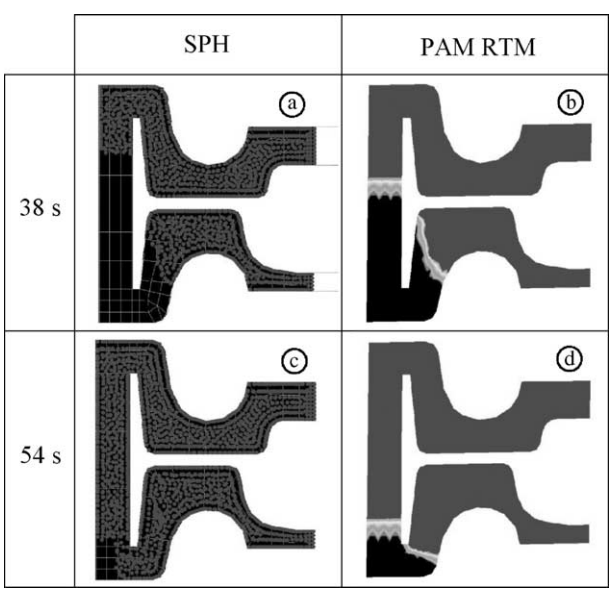

Fig. 2. Flow front position comparison for isotropic permeability filling simulations.

4. Experimental

4.1. Material characterization

An exhaustive materials characterization is performed at room temperature in order to feed the code with material input data. A non-reactive Newtonian fluid (Di-Octyl Phtalate, DOP) is used. Its viscosity, measured using a Brookfield viscometer, is 0.07 Pa s. The permeability of the woven fabric (1500S3, Chomarat Composites) is measured for fibre volume fractions varying from 0.43 to 0.57. The variation can be fitted to a second order polynomial as:

$$K = 1.08 \times 10^{-8} V_f^2 - 1.42 \times 10^{-8} V_f + 4.6 \times 10^{-9} \quad (10)$$

where V_f is the fibre volume fraction and the permeability K is in m^2. The areal weight of the fabrics is 1500 g m^{-2}. The foam core (NA1100, Alvéo) has also been characterized in compression. For the pressure levels used in this study, the foam can be considered as a linear elastic material and its bulk modulus is 1.2 MPa.

4.2. Sandwich manufacturing

Sandwich panels are manufactured at room temperature using a mould which cavity dimensions are 500×200×16 mm^3 (Fig. 3). The mould cavity is filled up with a 10 mm-thick soft closed-cell polyethylene foam core and two skins each consisting of 3 plies of glass woven fabric. The fibre volume fraction reached in both skins is close to 53%. Once the mould is closed, a fluid (DOP) is injected at constant flow rate (15 cl min^{-1}). Pressure at the injection gate ($P1$) and 29 cm away from the gate ($P2$) are monitored and recorded throughout the injection (Fig. 4). The pressure response at the gate location ($P1$ curve) can be separated into two regions: a rapid initial pressure increase during the first 25 s followed by a slower pressure increase. In the first

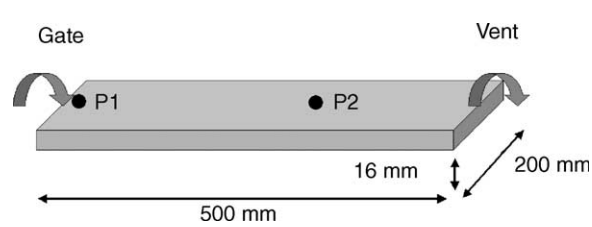

Fig. 3. Experimental setup.

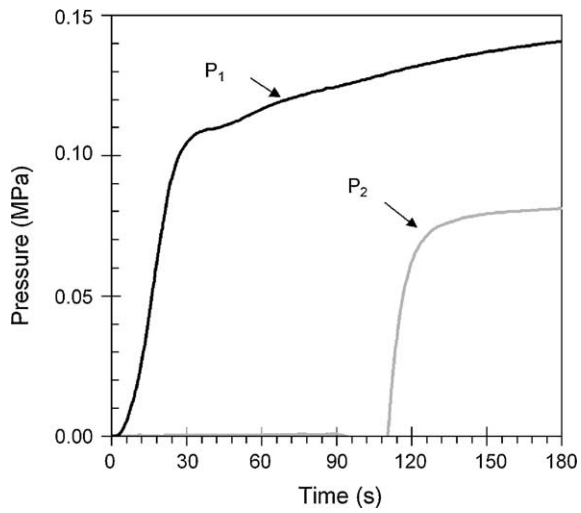

Fig. 4. Experimental pressure responses (P_1: at the gate. P_2: 29 cm from the gate).

region, the linearity between pressure and time verifies Darcy's law for unidirectional flow while the porous medium (fibre reinforcement) is not deformed. After 25 s of injection, foam core compression occurs due to a local pressure built-up in the mould cavity. In the neighbourhood of the gate, both porosity and permeability of the porous media increase, creating a discontinuity of slope of the pressure response—the resistance of the fluid to flow through the porous media decreases [3].

5. Simulation

In order to simulate the hydro-mechanical coupling observed experimentally, the mixed FE-SPH method is used. The foam core is modelled with FE as an elastic material, which is initially loaded under compressive stress. This initial compressive load is calculated from the equilibrium between the fibre reinforcement and the foam compression at the closing of the mould [3]. The fibre-reinforced skins are modelled as porous regions in which Lagrangian particles are evolving.

Special routines have been developed and implemented within PAM-CRASH to model as closely as possible the hydro-mechanical coupling. When the foam is being compressed, more volume becomes available to the flow of resin leading to expansion of the porous material (Fig. 5). When the foam deformations exceed a specific minimum value, a thin layer of free volume between the foam core and the porous material is created. Practical limitations do not allow to include this effect directly in the geometry of the computational model, but simple mathematical expressions are derived for the porosity and permeability as a function of the local core displacement. Therefore the user-defined routines (for porosity and permeability) are implemented and both parameters are calculated at each time step based

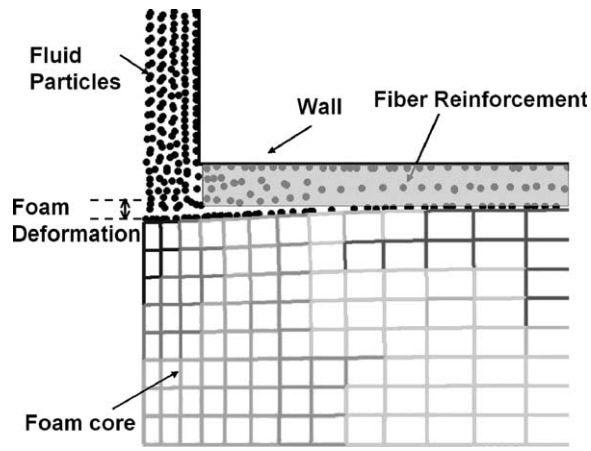

Fig. 5. Part of the domain used for the FE/SPH simulation. Grey scale of the mesh represents the stress levels.

on the local foam core compression (Fig. 6). As mentioned earlier, the fluid pressure response is transferred to the foam core using contact algorithms. Thus, no extra boundary conditions in between the porous and solid regions are required.

Fig. 7 shows the simulation results in terms of liquid pressure and foam core deformation at the injection gate. The pressure response increases with respect to time until the foam starts to deform ($t = 28$ s). After reaching a foam deformation of few tenths of millimetres, the pressure levels off. This behaviour simulated with the mixed FE-SPH method corresponds to the physics described experimentally. A comparison of the liquid pressure from the simulation with the experimental one also shows a quantitative good agreement. Fig. 5 is a close-up look at the foam deformation in the neighbourhood of the injection gate after 35 s of injection. Because of technical issues it was not possible to measure it experimentally, using a contact-less displacement sensor for instance, since the experiments were carried out in an industrial mould.

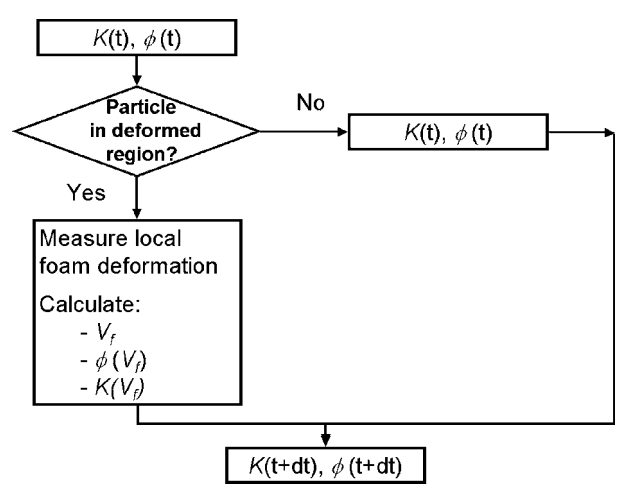

Fig. 6. Flow chart of the permeability K and porosity ϕ adjustments.

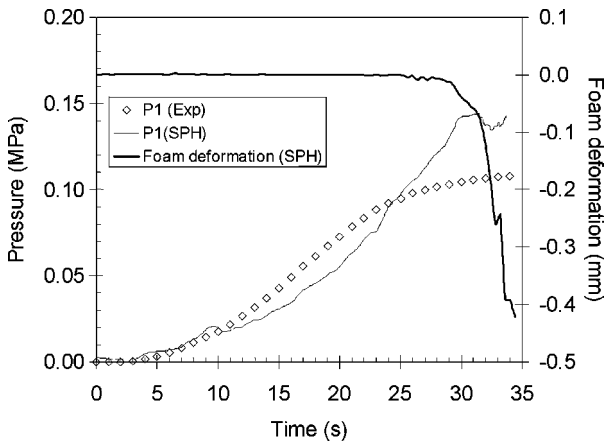

Fig. 7. Injection pressure and foam deformation.

However, the foam core deformation (few tenths of millimetres) and the pressure response effects obtained in this study from the simulation, are in the same range as those obtained by Binetruy et al. [3] and Wirth et al. [2] for RTM using very similar experimental conditions. Therefore, the accuracy of the method allows us to properly predict pressure and core crushing responses during the injection. Finally, the results of the study show that the simulation tool should be very sensitive. As a matter of fact, few tenths of millimetres of foam core crushing have a tremendous effect on the pressure response at the injection gate.

6. Discussion and conclusion

The pressure curve responses obtained from experiments and from simulation may differ because of inaccuracies of the physical input parameters (viscosity, foam bulk modulus, permeability…). For porous media flow according to Darcy's law, a typical 10% error on the measurement of the permeability of the fibre network will induce a 10% error on the pressure response for injections at constant flow rate. However, when hydro-mechanical coupling is relevant, the errors will propagate in a non-linear manner and will be quite difficult to estimate. Errors can also be present on the experimental curve since race-tracking and edge-effects may be present. The most important result of the simulation is that the liquid pressure response increases with respect to time until the foam starts to deform, then because of the permeability increase of the porous media, the liquid pressure reaches a plateau, as found in experiments.

In view of these uncertainties it was not found meaningful to conduct extensive testing of the sensitivity of the results on some numerical parameters (artificial viscosity, penalty coefficients of contacts, and coefficients of the Murnaghan equation). From previous studies on SPH and the coupling to finite elements, it is known that the influence on the results remains limited. The coupled SPH-FE solution is not expected to become a practical alternative to the existing Finite Element of Finite Difference solution for regular filling. However when coupling needs to be accounted, the SPH-FE method does provide an alternative. The main advantage of this method with respect to PAM-RTM and similar software is that it allows the interactive coupling with the structural deformation leading to a completely different flow because of permeability variations and the formation of channels (preferential flow) in between the fibre reinforcement and the foam for instance.

The mixed FE-SPH method has been used to solve for an example of hydro-mechanical coupling. A step toward the simulation of hydro-mechanical coupling has been reached using a very generic and versatile method. Another direct application would be to simulate the displacement of a foam core during sandwich manufacturing with RTM for instance. Also, the mixed FE-SPH method should have the potential to solve for non-isothermal fluid flows. Future developments would focus on the implementation of the equation of energy to simulate thermo-hydro-mechanical coupling in liquid composite moulding processes or include mechanical behaviour of fabrics for foam core shifting modelling for instance.

Acknowledgements

The authors would like to thank DGA (Délégation Générale pour l'Armement, France) for financial support for this study.

References

[1] Al-Hamdan A, Rudd CD, Long AC. Dynamic core movements during liquid moulding of sandwich structures. Composites Part A 1998;29A: 273–82.

[2] Wirth S, Gauvin R. Experimental analysis of core crushing and core movement in RTM and SRIM foam cored composite parts. J Reinf Plast Comp 1998;17(11):964–88.

[3] Binetruy C, Advani SG. Foam core deformation during liquid molding of sandwich: modelling and experimental analysis. J Sand Struct Mater 2003;5(4):351–76.

[4] Han K, Trevino L, Lee LJ, Liou M. Fibre mat deformation in liquid composite molding. I: experimental analysis. Polym Compos 1993; 14(2):144–50.

[5] Saunders RA, Lekakou C, Bader MG. Compression in the processing of polymer composites 1. A mechanical and microstructural study for different glass fabrics and resins. Compos Sci Technol 1999;59: 983–93.

[6] Batch GL, Cumiskey S, Macosko W. Compaction of fibre reinforcements. Polym Compos 2002;23(3):307–18.

[7] Pham X, Trochu F, Gauvin R. Simulation of compression resin transfer molding with displacement control. J Reinf Plast Comp 1998; 17(17):1525–56.

[8] Han K, Lee LJ, Liou M. Fibre mat deformation in liquid composite molding. II: modelling. Polym Compos 1993;14(2):151–60.

[9] Farina A, Cocito P, Boretto G. Flow in deformable porous media: modelling and simulations of compression moulding processes. Math Comput Modelling 1997;26(11):1–15.

[10] Sawley M, Cleary P, Ha J. Modelling of flow in porous media and resin transfer moulding using smoothed particle hydrodynamics. In: Second international conference on CFD in minerals and process industries, CSIRO, Melbourne, Australia 6–8 December 1999 p. 473–8.
[11] Tucker III CL, Dessenberger B. Governing equations for flow and heat transfer in stationary fibre beds. In: Advani SG, editor. Flow and rheology in polymer composite manufacturing. Amsterdam: Elsevier; 1994. p. 257–323.
[12] Pillai KM. Governing equations for unsaturated flow through woven fibre mats. Part 1. Isothermal flows. Compos Part A 2002;33(7): 1007–19.
[13] Groenenboom PHL. Application of the mixed SPH-FE approach in PAM-CRASH to deformable porous media. ESI BV Internal report; 2001.
[14] Lucy LB. A numerical approach to the testing of fusion process. Astron J 1977;88:1013–24.
[15] Gingold RA, Monaghan JJ. Smoothed particle hydrodynamics: theory and application to non-spherical stars. Mon Not R Astr Soc 1977;181: 375–89.
[16] Monaghan JJ. Simulating free surface flows with SPH. J Comput Phys 1994;110:399–406.
[17] Schlatter B. A pedagogical tool using smoothed particle hydrodynamics to model fluid flow past a system of cylinders. Dual MS project, Oregon State University; June 1999.
[18] Hyncik L. Lagrangean approach to flow simulations by smoothed particle hydrodynamics. In: Proceedings of the UWB, Plzen, Czech Republic. vol. 1 2000.
[19] Shao S, Lo EYM. Incompressible SPH method for simulating Newtonian and non-Newtonian flows with a free surface. Adv Water Resour 2003;26(7):787–800.
[20] Pentecote N, Kohlgrueber D, Kamoulakos A. Simulation of water impact problems using the smoothed particle hydrodymanics method. In: ICD'03 conference, Lille, France December 2003.
[21] PAM-CRASH Notes Manual. ESI-Group Trademark; 2001.
[22] Fulk DA. A numerical analysis of smoothed particle hydrodynamics. PhD Thesis, Air Force Institute of Technology; 1994.
[23] Monaghan JJ, Gingold RA. Shock simulation by the particle method SPH. J Comput Phys 1983;52:374–89.
[24] Meywerk M, Decker F, Cordes J. Fluid-structure interaction in crash simulation. Proc Inst Mech Engrs 1999;214:669–73.
[25] Cha H, Lee I, Choi HY. Industrial applications of PAM-SHOCK using SPH. In: PAM users conference in Korea HANPAM'99. Seoul 15–16 November 1999 p. 253–65.
[26] Haack C. On the use of a particle method for analyis of fluid–structure interaction. Sulzer Innotech Report. STR_TB2000_014 June 2000.
[27] Gauvin R, Trochu F, Lemenn Y, Diallo L. Permeability measurement and flow simulation through fiber reinforcement. Polym Compos 1996;17:34–42.