VII International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2017
M. Papadrakakis, E. Oñate and B. Schrefler (Eds)

COUPLING MULTIPHYSICS PROBLEMS IN TRANSIENT REGIMES: APPLICATION TO LIQUID RESIN INFUSION PROCESS

RABIH MEZHER*, SIMON BREUMIER, NICOLAS MOULIN, JULIEN BRUCHON, AND SYLVAIN DRAPIER

SMS Division and LGF UMR CNRS 5307
Mines Saint-Etienne - Université de Lyon
158, Cours Fauriel, CS 62362, 42023 Saint-Etienne cedex 2, France
e-mail: *rabih.mezher@emse.fr, simon.breumier@etu.emse.fr, nmoulin@emse.fr, bruchon@emse.fr and drapier@emse.fr

Key words: Resin Infusion, Composite Materials, Fluid-Solid Coupling, Weak Coupling, Message Passing Interface

Abstract. Liquid resin infusion (LRI) process is widely considered in the aeronautics, due to its benefits (low void content and production of large parts), for high performance composite material forming. The main objective of the present work is to simulate numerically the LRI process, in a high performance computing framework, which consists in coupling fluid-solid mechanics. Hence, two fluid flow regimes are coupled with an efficient ASGS stabilized monolithic finite element formulations: the resin flow in both a highly permeable distribution medium (Stokes) and low permeability fibrous orthotropic preforms (Darcy). Moreover, weak coupling algorithms are used along for coupling solid / fluid mechanics, solid / level-set problems and fluid / level-set problems; where the level-set method is used to capture the moving flow front and the Stokes-Darcy interface. To transfer the different physical variables between the above coupled problems, Message Passing Interface (MPI) library is chosen, to ensure the best data transfer performances.

1 INTRODUCTION

In today's aeronautical and automotive industry, composite materials have become the primary choice in the manufacturing processes, due to their light weight, and overall properties (mechanical, thermal, electrical ...). The Liquid Resin Infusion (LRI) processes allow the manufacturing of high performance composites and are among today's competitive techniques, because they can achieve very low void contents, on large pieces and particularly on aircraft parts [1]. In LRI processes, a dry fiber reinforcement is placed (Figure 1) into a mould, and then enclosed in a specific bagging system. Vaccum is then exerted on the reinforcement, leading to an atmospheric compression of the reinforcement. Hence under this depression, a liquid thermoset resin fills in a stiff distribution medium placed under or over the preform to create a resin-feeding bed. As a result, the resin flows through the layer stacking that swells simultaneously due to the fluid resin pressure effects [2]. When the resin fully impregnates through the reinforcement, the supply of resin is cut off and resin is left for curing, still under vacuum pressure. Even though LRI processes present many advantages, they are difficult to control experimentally. When critical properties are involved in such processes (dimensions of the final pieces, fiber volume fraction, ...), the control becomes even more challenging. Therefore, it is necessary to use numerical simulations in order to control and properly understand the interactions of the fibrous preform non-linear deformations with the resin flow, out of (Stokes flow regime) and in (Darcy flow regime) these orthotropic deformable porous media. The objective of this paper is to present a robust numerical approach, able to couple the different complex problems involved in LRI processes. To assure multiphysic coupling, a suitable coupling method has to be chosen, ensuring the transfers of the different physical variables between the problems. Moreover, an algorithm is later selected to couple the different problems. This paper is structured as follows: first coupled problems are identified and detailed; then the coupling techniques are presented, based on the MPI library [3] for transfering variables between problems; later the algorithm to couple fluid and non-linear solid mechanics is exposed; finally the results of the numerical simulations of the coupled problems featured in resin infusion are shown.

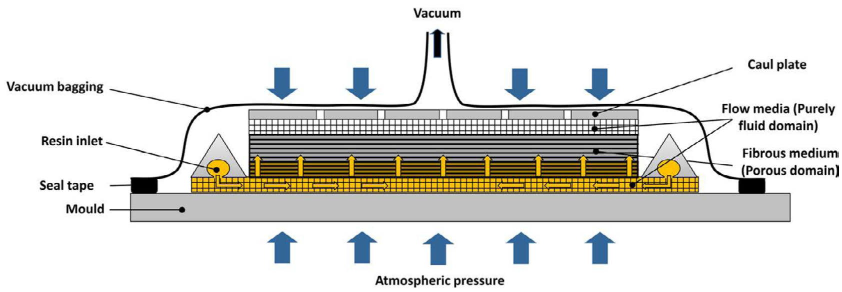


Figure 1: Schematic principle of Liquid Resin Infusion process [2]

2 COUPLED PROBLEMS

Modeling the LRI processes consists in coupling fluid, solid, and porous mechanics in isothermal conditions (Figure 2). First, two fluid flow regimes are coupled, representing the resin flow in both a highly permeable distribution medium (Stokes) and low permeability fibrous orthotropic preforms (Darcy). Darcy system expresses the flow velocity as a function of the pressure gradient and two parameters, the fluid viscosity η and the permeability $\mathbf{K}$ which traduce the ability of the media to be penetrated by the fluid. The problem is then desribed by finding a velocity field $\mathbf{v}_d$ and a pressure field p_d such that :

$$\begin{aligned} \eta \mathbf{K}^{-1} \mathbf{v}_d + \boldsymbol{\nabla} p_d = \mathbf{f}_d \\ \boldsymbol{\nabla} \cdot \mathbf{v}_d = 0 \end{aligned} \tag{1}$$

with $\mathbf{f}_d$ the external forces. Stokes problem expresses the velocity as a function of the pressure gradient and the viscosity; it is written as follows:

$$\begin{aligned} -\boldsymbol{\nabla} \cdot (2\eta \dot{\varepsilon}(\mathbf{v}_s)) + \boldsymbol{\nabla} p_s = \mathbf{f}_s \\ -\boldsymbol{\nabla} \cdot \mathbf{v}_s = 0 \end{aligned} \tag{2}$$

with $\mathbf{f}_s$ the external forces; and $\dot{\varepsilon}(\mathbf{v}_s) = \frac{1}{2}\left(\boldsymbol{\nabla}\mathbf{v}_s + \boldsymbol{\nabla}^T\mathbf{v}_s\right)$, is the strain rate. This Stokes-Darcy coupling is numerically achieved with an efficient ASGS stabilized monolithic finite element formulation (for the detailed description the reader can refer to [1]). For the solid problem, the general formulation is written in the framework of an updated Lagrangian formulation [4, 5, 6]:

$$\boldsymbol{\nabla} \cdot \boldsymbol{\sigma} + \mathbf{f} = \mathbf{0} \tag{3}$$

where $\boldsymbol{\sigma}$ is the Cauchy stress tensor and $\mathbf{f}$ are the external forces. Hence, the displacement fields are obtained by solving equation (3). Solid and fluid interact in two ways in the wet preforms, since the resin pressure exerted by the fluid will deform the preforms and lead to a change in their porosity/permeability. In turn this will affect the fluid flow. The preforms exhibit swelling under the effect of the fluid's pressure ; this is modeled by using Terzaghi's equivalent stress. Note that, the dry preforms are compacted, because they are subjected to the external atmospheric pressure; this later results in displacements of the preforms. Last, the level-set approach used to capture the flow front will rely on the physical fluid velocity and on the mesh velocity (i.e. the mesh of the domain is deformed because of mechanical displacements). The level-set function (denoted by ψ) is transported according to the hyperbolic equation:

$$\frac{\partial \psi}{\partial t} + \mathbf{v} \cdot \nabla \psi = 0 \tag{4}$$

where $\mathbf{v}$ is the velocity computed from the Stokes-Darcy problem.

3 COUPLING TECHNIQUES

To perform multiphysical coupling, two approaches can be considered: a monolithic method and a weak coupling based method. The monolithic approach considers the coupled problems as a single block, hence solving simultaneously for two or more problems; this type of approach denotes a strong coupling [2, 9] and is used to couple the Stokes-Darcy problem reported in section 2. The weak coupling based method, consists of seperately solving each problem iteratively, until reaching the global solution. This type

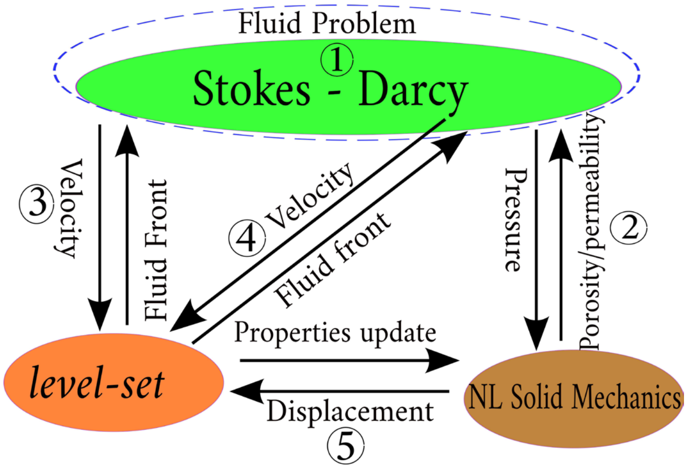


Figure 2: Schematic representation of the coupled problems

of approach is proposed to couple the problems illustrated in Figure 2. Transfering data and variables between the coupled problems, requires the usage of a suitable exchange library. The main objective is to minimise the computation time (and particularly the variable exchange time), when these high performance calculations are done. Within a parallel computing framework, each physical problem can be solved on a processor accessing only to its own memory. In this case, the MPI (Message Passing Interface) library is used for the transfer of the different required data [7].

4 COUPLING ALGORITHM

Non-linear solid mechanics, fluid mechanics and level-set problems are weakly coupled with a sequential algorithm (Algorithm 1). Subsequently, at each time step until the preform is filled, three problems are solved.

Figure 3 details the process of exchanging variables at each time step (Δt being the time step and t denotes the time), showing which variables are sent (respectively received by each code or problem). In what follows, and for the sake of clarity the different problems will be numbered according to Figure 3. Figure 4 shows the global resolution of the problems at each time step. Before the solving process, an initialisation of the Stokes-Darcy interface (denoted by $\Gamma_{s,d}$) and of the initial position of fluid flow front (denoted by Γ_f) is done. Then the fluid problem transfers its velocity ($\vec{v}$) to the level-set problem. After that, the solid problem receives the fluid pressure and the position of the fluid front (i.e. the level-set function value ψ), thus updating the effective stress $\boldsymbol{\sigma}(\boldsymbol{\epsilon})$ according to Terzaghi's law [8]; then the solid problem iterates til convergence. Finally the mesh is updated (this is done at each time step), taking into account the compaction of the preforms ($\vec{u}$ denotes the displacements).

Algorithm 1 Sequential algorithm

Begin of weak coupling: Initialisation of the Stokes-Darcy interface $\Gamma_{s,d}$ and the fluid flow front Γ_f
do:

1. Computation of the preform compaction while considering the fluid pressure with Terzaghi's law (displacements), then update mesh
2. Computation of the Stokes-darcy fluid problem (velolcity and pressure)
3. Computation of the resin flow front motion: resolution of the level-set problem

until the preform is filled ($t + \Delta t$)
End of weak coupling

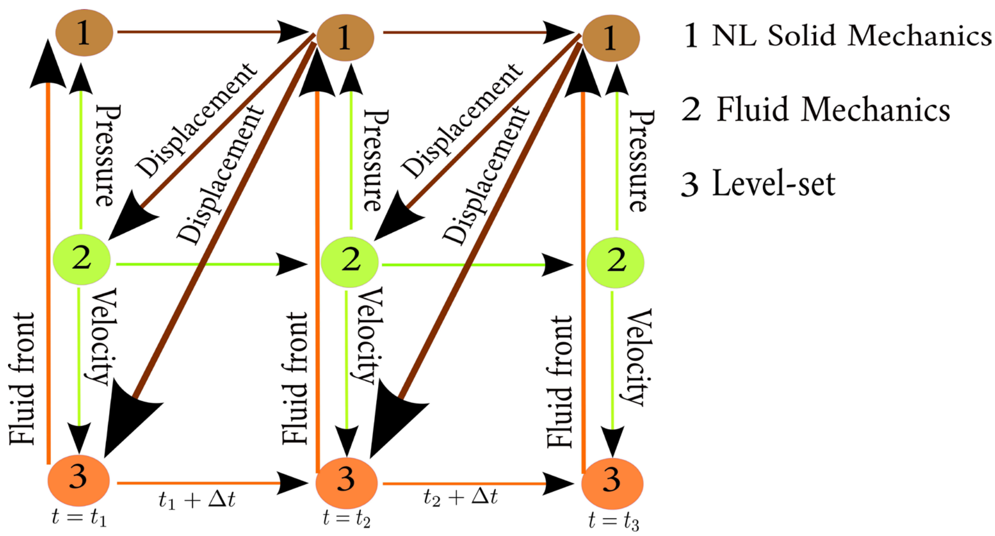


Figure 3: Sequential algorithm with the designated sent and received variables between the three problems (NL is the abbreviation for non-linear)

5 NUMERICAL SIMULATIONS

5.1 Geometry and boundary conditions

The coupling between problems 1, 2, and 3 is simulated on a "T" shaped geometry. This shape represents a part used in aeronautical engineering (stiffened panel). For the Stokes-Darcy problem, one single unstructured mesh is used. Stokes and Darcy subdomains (Figure 5) are subsequently described by using a level-set function. The dimensions of the part are shown in Figure 5, where symmetry conditions are considered. The Stokes-Darcy interface is illustrated by the zero isosurface value coloured in red on Figure 5.

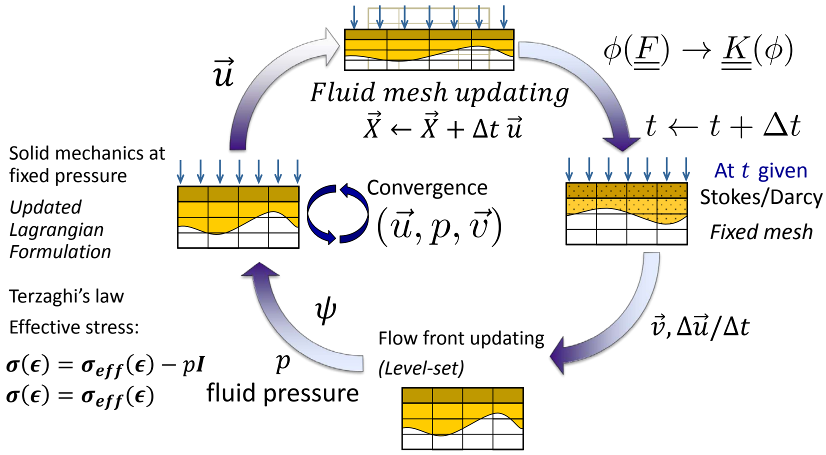


Figure 4: Global time-stepping strategy for solving the coupled problems [2, 5, 9]

A pressure gradient of 10^5 Pa is applied between top and bottom faces of the domain (Figure 5 where σ_n is the normal stress). To represent impervious walls, normal velocities are considered equal to zero (i.e. $\vec{v} \cdot \vec{n} = 0$ on Figure 5, where $\vec{v}$ is the fluid velocity and $\vec{n}$ is the normal vector to the domain boundary), on the remaining boundaries.

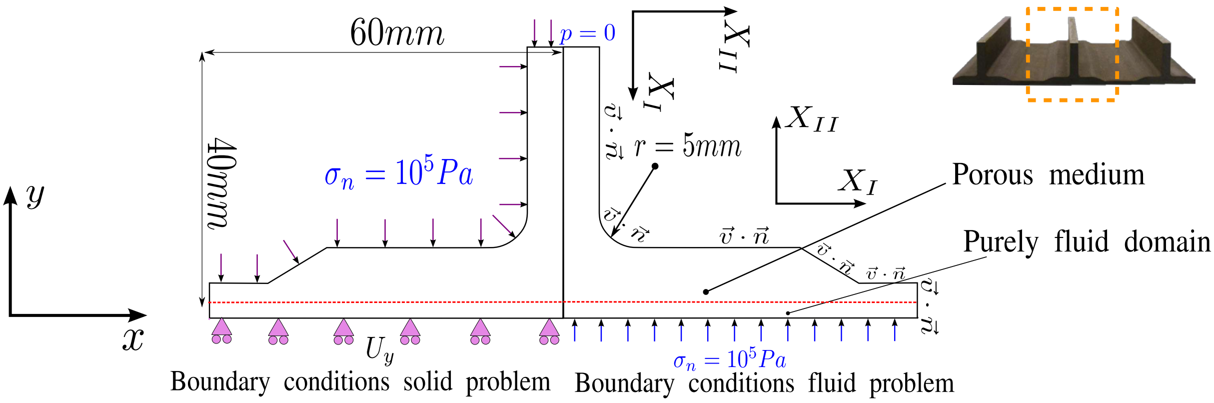


Figure 5: Boundary conditions for both the fluid and solid problem (r is the radius of the curvature) [1]

For the non-linear solid mechanics problem, displacement conditions are inforced as shown on Figure 5. They represent the contact of the preform with the mould, where U_y denotes the components of the deplacements in the y direction and U_x denotes the deplacements in the x direction ($\vec{U} = U_x.\vec{x} + U_y.\vec{y}$). A pressure of 10^5 Pa is applied on the rest of the boundaries, which is the atmospheric pressure on the vaccum bag.

5.2 Numerical Results

The coupling between the three problems, has been done by using the finite element software Zset [10]. The computations were carried out on one multiprocessor computer; hence, each problem runs on one processor, of the same machine. MPI library was used to handle the exchange in data between codes, in conformity with the scheme presented in Figure 3. Five numerical simulations were executed on five unstructured meshes (denotes by meshes 1, 2, 3, 4 and 5 in Table 1) of linear triangles. The time step used for all simulations was $\Delta t = 0.05\,s$ (sufficiently small in order to reach convergence) for a final physical time of $t = 4\,s$. The mesh size (i.e. number of nodes and elements), as well as the parameters used for the fluid problem (viscosity η, permeabilities K_I and K_{II}; $\mathbf{K} = K_I\ \vec{X}_I \otimes \vec{X}_I + K_{II}\ \vec{X}_{II} \otimes \vec{X}_{II}$) and the solid problem (Young's modulus E, poisson's coefficient ν) are summarized in Tables 1 and 2. In order to emphasize on the performance of the weak coupling using MPI library, we considered only one viscosity on the hole domain (for a more richer description the reader can refer to [1]). Figure

Mesh	Number of nodes	Number of elements
1	1106	2212
2	2060	4120
3	3063	6126
4	12104	23544
5	26726	52462

Table 1: Mesh size parameters

K_I (m^2)	K_{II} (m^2)	η $(Pa.s)$	E (MPa)	ν
100.10^{-11}	10^{-11}	0.03	210	0.3

Table 2: Parameters for both the fluid and solid problem

6(a) shows the pressure field after the compaction of the preforms under the effect of the atmospheric pressure. Once the preforms are compacted due to the external atmosperic pressure, the mesh geometry is updated. Figure 6(b), shows the velocity field magnitude.

Figure 7(a) shows the displacements U, where the original mesh (coloured in grey) has

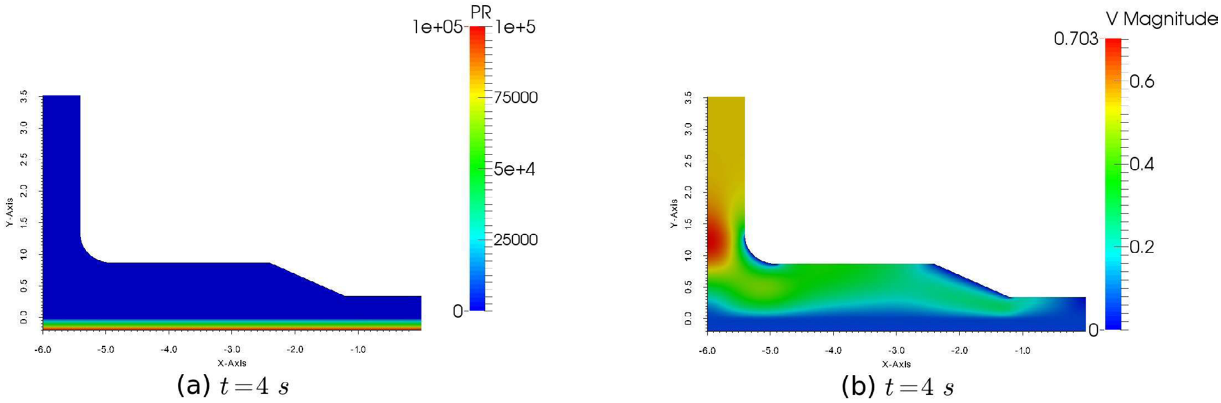
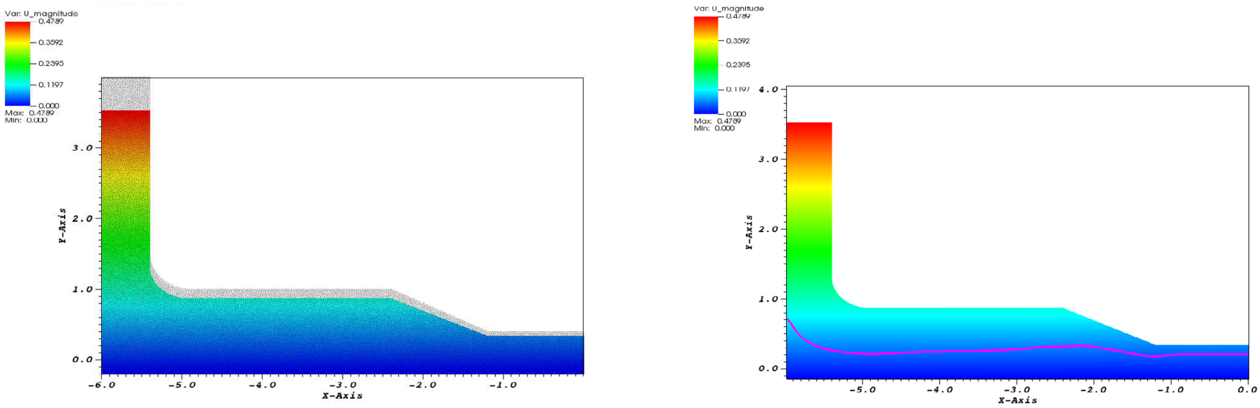


Figure 6: Pressure and velocity fields

been superimposed on the deformed mesh, in order to show the preform's compaction. The movement of the fluid flow front (pink line) is also depicted on Figure 7(b). The grey mesh in Figure 8 corresponds to the initial undeformed mesh (before compaction) superimposed on the deformed mesh (i.e. the coloured mesh is the deformed one). It has been represented seperately here from the physical fields, in order to evaluate quantitatively and qualitatively the large deformartions exhibited by the preforms. It is noticebale from Figure 8, that deformations are not homogeneous on the global domain; this is mainly due to the non-homogeneous pressure field owing to Terzaghi's law and anisotropic permeabilities. Two points of interest (points A and B on Figure 8) have been chosen to discuss and emphasize on the thickness variation. The cross section in point A is found to sustain approximately 13% (Table 3, where H_A denotes the height in point A) of thickness variation, while the one in point B undergos 10% (Table 3, where H_B denotes the height in point B) of thickness variation. The numerical values of computation time is shown in Table 4. The computation time (denoted by CT in Table 4) increases with the number of degrees of freedom associated with the sent / received variables. Hence, more time is needed to assure the exchange of much more volumetric data. This evolution is compared with a previous exchange method adopted in [1]. It is clearly shown from Figure 9 (where the computation time of the two methods is shown), that the computation time is much higher with this last technique. Therefore, MPI allowed a significant reduction and optimization in computation time (the percentage of time reduction is shown in Table 4), relative to this classical technique, and especially for high mesh refinements (for 26726 nodes, the computation time is approximately reduced by a factor of 2).

6 CONCLUSION

Starting from previous works done by [1, 2, 4, 9], a robust approach has been proposed in this work to couple monolithic Stokes-Darcy fluid mechanics, non-linear solid mechan-



$(a)\, t = 1\, s$ $(b)\, t = 4\, s$

Figure 7: Displacement field magnitude: (a) original non deformed mesh (coloured in grey) superimposed on the deformed geometry and (b) the change in the moving fluid flow front (pink line)

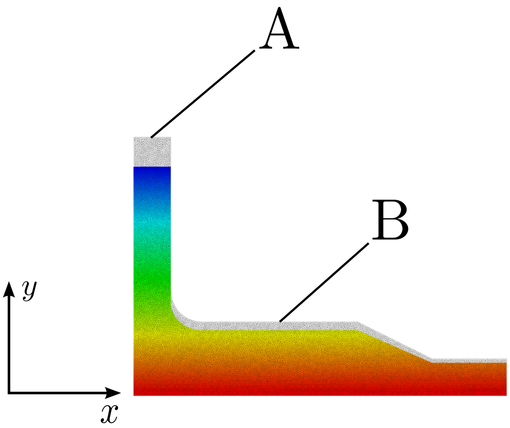


Figure 8: Mesh geometry before and after compaction

Height before compaction	Height after compaction	Percentage of thickness variation
$H_A = 4\,cm$	$H_A = 3.5\,cm$	12.5%
$H_B = 1.1\,cm$	$H_B = 1\,cm$	10%

Table 3: Thickness variations in point A and B in Figure 8

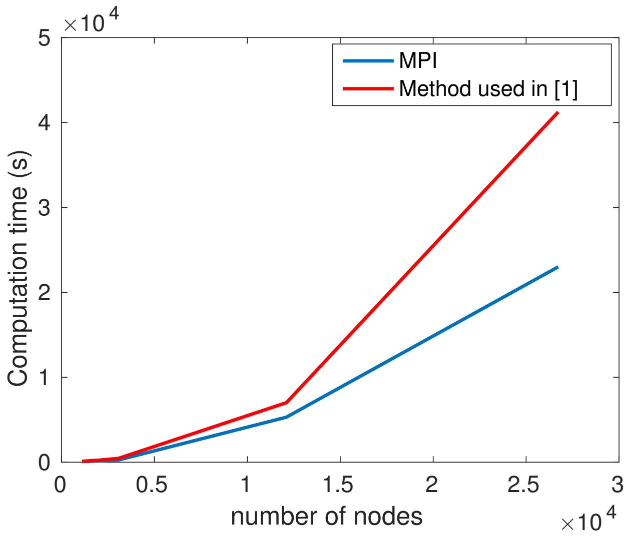


Figure 9: Computation time for both data exchange methods: MPI and method used in [1]

Mesh	Number of nodes	CT (MPI)	CT (method used in [1])	Percentage of time reduction
1	1106	74	104	29%
2	2060	148	245	40%
3	3063	254	442	43%
4	12104	5300	7006	24%
5	26726	23000	41240	44%

Table 4: Numerical values of computation time for both data exchange methods: MPI and method used in [1]

ics and level-set transient problems. The coupling was performed on a complex industrial piece, reported in previous works . It was shown that the coupling of the fluid and solid problems has an effect on the final dimensions of the piece. Moreover, MPI was used to ensure data transfer between distant codes where high performance computation were done on refined meshes. The computation time has been optimized with MPI, in comparison with classic conventional techniques. Future works, should focus on integrating more problems to the couplings, such as temperature and thermo-chemical effects (involved in resin curing). In addition, parallel computations have to be considered for more refined meshes; this is currently a work in progress.

REFERENCES

[1] Blais, M. and Moulin, N. and Liotier, P.J. and Drapier, S. Resin infusion-based processes simulation : coupled Stokes–Darcy flows in orthotropic preforms undergoing finite strain. *Int. J. Mater. Form.* (2017) **10(1)**:43–54.

[2] Abou Orm, L. and Troian, R. and Drapier, S. and Bruchon, J. and Moulin, N. Stokes–Darcy coupling in severe regimes using multiscale stabilisation for mixed finite elements : monolithic approach versus decoupled approach. *European Journal of Computational Mechanics.* (2014) **23**:113–137.

[3] Gropp, W. and Lusk, E. and Skjellum, A. *Using MPI Portable Parallel Programming with the Message Passing Interface.* third edition, scientific and engineering computation series (2014).

[4] Celle, P. and Drapier, S. and Bergheau, J.M. Numerical aspects of fluid infusion inside a compressible porous medium undergoing large strains. *Revue européenne de mécanique numérique. Resin Infusion-Based Processes Simulation.* (2008) **17(5–6–7)**:819–827.

[5] Celle, P. and Drapier, S. and Bergheau, J.M. Numerical modelling of liquid infusion into fibrous media undergoing compaction. *Eur. J. Mech. A. Solids.* (2008) **27(4)**:647–661.

[6] Dereims, A. and Drapier, S. and Bergheau, J.M. and de Luca, P. 3d robust iterative coupling of Stokes, Darcy and solid mechanics for low permeability media undergoing finite strains. *Finite Elem. Anal. Des.* (2015) **94**:1–15.

[7] Garaud, J. D. Développement de méthodes de couplage aéro-thermo-mécanique pour la prédiction d'instabilités dans les structures aérospatiales chaudes. PhD Thesis. (2008)

[8] Terzaghi, K. *Soil mechanics in engineering practice.* John Wiley and Sons, (1996).

[9] Pacquaut, G. and Bruchon, J. and Moulin, N. and Drapier, S. Combining a level-set method and a mixed stabilized P1/P1 formulation for coupling Stokes–Darcy flows. *Int. J. Numer. Meth. Fluids.* (2012) 459–480.

[10] Besson, J. and Foerch, R. Large scale object-oriented finite element code design. *Comput. Methods Appl. Mech. Eng.* (1997) **142(1)**:165–187.