A FLUID-STRUCTURE INTERACTION APPROACH FOR THE SIMULATION OF MOULD FILLING USING THE LIGHT-RTM TECHNIQUE

S. Kanarachos*, N. Pantelelis

[1]*Frederick Research Center, Mechanical Engineering Department, Frederick University, 7 Y. Frederickou St., Lefkosia, Cyprus*

**eng.ks@frederick.ac.cy*

Keywords: Light-RTM, mould flexibility, fluid-structure interaction, mould filling, dimensional accuracy

Abstract (Times New Roman 12 pt, bold, single-line spacing, left-aligned text)

In this study a fluid-structure interaction approach is proposed for the simulation of mould filling using the Light-RTM technique. Light-RTM is an economical production method which utilizes moulds with one rigid and one semi-rigid part. The semi-rigid part deforms during the mould filling process as a result of the force balance between the atmospheric pressure outside the mould and the resin pressure inside the mould. The deformation of the flexible cover, which depends on resin pressure and mould's shape, affects the resin flow front during mould filling and the dimensional accuracy of the final part. Coupling between the mechanical deformation of the flexible cover and the resin flow inside the mould cavity is achieved by implementing an indirect fluid-structure interaction approach using ANSYS software. Based on this model, a general methodology is developed to simulate numerically the resin flow during the mould filling process. A number of case studies have been simulated using the proposed methodology.

1 Introduction

Light-Resin Transfer Molding (Light RTM) is a variant of the Resin Transfer Molding technique and belongs to the closed mould composite production methods. A closed mould consisting of two parts and made from fiberglass encloses the dry perform before the infusion begins. A resin pump is utilized to suck the unsaturated resin from a storage medium and infuses it under pressure inside the -initially unfilled- mould cavity. Under the action of the differential pressure, between the mould inlets and outlets, the resin progresses and gradually fills the mould until the entire perform is completely wet out. At this point the infusion process stops and sufficient time is given for the resin to saturate and become a solid part [1].

Light RTM belongs to the economical production methods. Its distinctive characteristic is that one side of the mould- the upper one- is semi-rigid. In many cases the resin flow under the cover is visible (Figure 1). It is not as flexible as in the case of the vacuum bag and not as rigid as in the case of RTM. The main driver for constructing the upper mould part semi-rigid is to reduce the mould manufacturing cost in cases in which the production size is medium to small. The optimum production size for the Light RTM technique has been specified to be between 50 and 500 parts however deviations exist depending on the specific features of the part to be manufactured [2], [3]. During the mould filling process the semi-rigid mould deforms as a result of the force balance between the atmospheric pressure outside the mould and the resin pressure inside the mould. The deformation of the flexible cover, which depends on resin pressure and mould's shape, affects the resin flow front during mould filling as well as the dimensional accuracy of the final part (see Figure 1) [4]. It is highlighted that the dimensional accuracy and final composite part quality is dependent on the mould's deformation not only at the end of the filling process but also during it, due to the high relaxation times which can last up to 10 minutes or even more [5], [6], [7], [8].

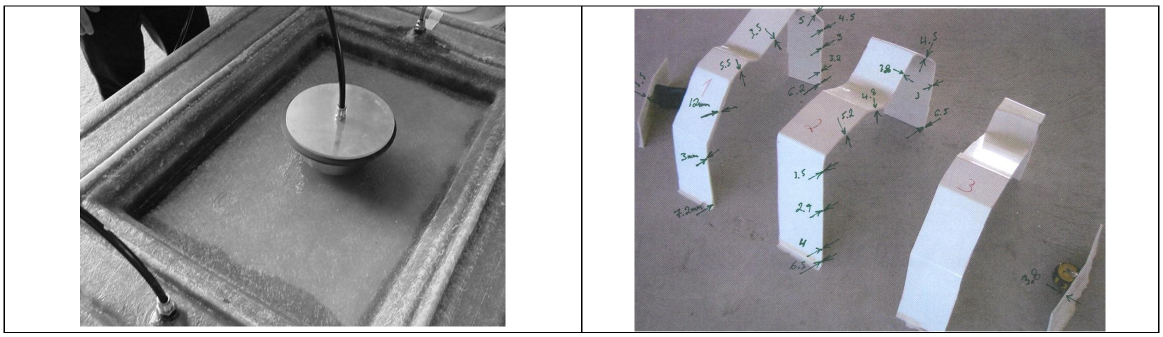

Figure 1. Left part: Typical Light-RTM moulds with their ancillaries **Right part:** Thickness deviations resulting from mould deformation

In this study a fluid-structure interaction approach is proposed for the simulation of mould filling using the Light-RTM technique. Coupling between the mechanical deformation of the flexible cover and the resin flow inside the mould cavity is achieved by implementing an indirect fluid-structure interaction approach using ANSYS software [9]. Based on this model, a general methodology is developed to simulate numerically the resin flow during the mould filling process. According to our knowledge this is the first time such an approach is followed and presented. A number of case studies have been studied using the proposed methodology and two of them will be presented in this paper.

The rest of the paper is organized as follows. In Chapter 2 the mathematical model used for simulating the resin flow in the mould cavity is described. In addition, a description of the coupling between the fluid and structure numerical results is given. In Chapter 3 the numerical model and a description of the solution methodology is given. In Chapter 4 numerical results for two case studies are presented and discussed. In Chapter 5 conclusions are drawn and future research directions are given.

2 Mathematical model

Two distinct multiphase flow models are available; the Euler – Euler multiphase model and the Lagrangian Particle Tracking multiphase model. In the present work, the flow of resin and air in the mould was modeled using the Eulerian approach due to its obvious computational appropriateness at free surface flows.

2.1 The Euler-Euler Flow Equations

The motion of fluids is expressed by conservation laws of mass, momentum and energy. The equation for mass is known as the continuity equation while the equation for momentum is called equation of motion as it is an expression of Newton's law. If viscous fluid and inviscid fluid are considered in these equations, they are known as the Navier-Stokes and Euler equations respectively. In the general form the set of governing equations for homogeneous mixture of multiphase flows in the Cartesian coordinates are as follows:

$$\frac{\partial \rho_m}{\partial t} + \nabla(\rho_m \cdot U_m) = 0 \qquad (1)$$

$$\frac{\partial}{\partial t}(\rho_m \cdot U_m) + \rho_m \cdot \nabla(U_m \cdot \nabla)U_m = -\nabla(p_m) + \nabla(\tau + \tau_t) + f \qquad (2)$$

where p is pressure, τ is the viscous stress tensor, τ_t is the turbulent viscous stress tensor and f is the buoyancy force.

For multiphase flows a volume fraction transport equation is added to the governing equations as follows:

$$\frac{\partial}{\partial t}(\alpha_i \cdot \rho_i) + \nabla(\alpha_i \cdot \rho_i \cdot U_m) = 0 \qquad (3)$$

where α_i represents the liquid phase volume fraction.

Turbulence equations are not considered as the flow in the mould cavity is laminar.

2.2 Flow in porous media

The volume porosity γ at a point is the ratio of the volume V' available to flow in an infinitesimal control cell surrounding the point and the physical volume V of the cell:

$$\gamma = \frac{V'}{V} \qquad (4)$$

The porosity is taken intro account in the mass and momentum conservation equations. Furthermore, the pressure drop for a fluid moving through a porous media is described in the form of Darcy's law which is given by

$$\nabla(p_m) = \frac{\mu}{K_{perm}} \cdot U_m + K_{loss} \cdot \frac{\rho}{2} \cdot |U| \cdot U_m \qquad (5)$$

where μ is the dynamic viscosity, K_{perm} is the permeability and K_{loss} an empirical factor.

2.3 Fluid-structure interaction

In fluid-structure interaction problems the boundaries of the fluid domain are not considered stationary anymore. Instead they can move and deform. In order to account for the arbitrary movement of the grid, it is necessary to modify the governing discrete equations to include the motion of the grid as well as the fluid in the transient convection terms. If this is not done, a number of artefacts can result, including grid-generated flows, when a grid is moved through a stationary flow. Therefore, after the boundary co-ordinates are updated the interior points of the finite element mesh are recalculated. Finally, the governing mass and momentum equations are formulated in the resulting fluid domain.

Regarding the coupling of the fluid and structural results a simple file-based approach is currently used. More specific, the Computational Fluid Dynamics (CFD) software is writing out a 'results' file. This file is a binary direct access file, in a compressed format, containing selected results (e.g. pressure field). This file is picked up by the structural analysis program, which carries out the calculation of the forces and the interpolation. Finally, the structural analysis software updates the solution of the problem and transfers the mesh motion to the CFD software.

3 Numerical solution – CFD and structural analysis software

For the simulation of the resin flow in the mould cavity the software *Ansys CFX 11.0* has been used. For the simulation of the structural response of the mould the *Ansys Mechanical* software has been utilized.

3.1 Ansys CFX

CFD is a tool based on a computer for simulating the behavior of systems involving fluid flow, heat transfer and other processes. The accurate prediction of flow and the pressure field in the mould provides the opportunity to determine a priori with significant accuracy the deformation of the flexible mould and as a consequence the final dimensional accuracy of the produced part. 3D numerical software is used to solve such kinf of problems. This has been made possible by advances in computer technology and numerical algorithms. One of the most advanced CFD codes is *Ansys CFX 11.0* software. *Ansys CFX 11.0* is a code that used the finite volume approach for solving the complete Reynolds averaged Navier-Stokes equations. The numerical solution of the continuity and momentum equations is obtained using the CFD code *Ansys CFX 11.0*, which is a vertex-centered code based on the finite volume numerical method. The Navier-Stokes conservation equations described above are discretized using an element-based finite volume method [9], [10]. The discretization of the conservation equations is fully conservative and time-implicit. The conservation equations are integrated over each control volume, volume integrals are converted to surface integrals using Gauss' divergence theorem, and surface fluxes are evaluated in exactly the same manner at the two control volumes adjacent to an integration point.

The advection scheme used to evaluate the variable (φ_{ip}) in terms of neighboring vertex values (φ) is extremely important for the solution accuracy. It can be written as

$$\varphi_{ip} = \varphi_{up} + \beta \cdot \nabla \varphi \cdot \Delta r \tag{6}$$

φ_{up} is the upwind vertex value and Δr is the vector from the upwind vertex to the integration point. The quantity $\beta \cdot \nabla \varphi \cdot \Delta r$ is called numerical advection correction. If β= 0, this scheme recovers the first-order upwind scheme, which is bounded but excessively diffusive. If β= 1, this scheme is a second-order upwind-biased scheme, but unbounded. A bounded high resolution scheme, used in this work, can be obtained by making β as close to 1 as possible, but reducing where necessary to prevent overshoots and undershoots from occurring. For standard advection terms, *CFX* uses a method similar to that described by [11]. This numerical method is formally second order accurate as well as bounded since it reduces to first order only near discontinuities. Accuracy and stability are then assured, while non-physical numerical oscillations are avoided. The mass flows must be discretized carefully to avoid pressure–velocity decoupling. This is performed by generalizing the interpolation scheme proposed by [12]. The linear system of equations is solved using a coupled algebraic multigrid technique [10].

3.2 Ansys Mechanical

Ansys Mechanical is a general purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electromagnetic problems.

4 Numerical results – Case studies

In this paper two numerical case studies will be presented. They concern two flat rectangular plates with areas of 1000*1000 mm^2 and 500*500 mm^2 respectively. The thickness of the plates and its effect on the deformation of the mould during and at the end of the filling process is the focus of a parametric analysis. We will show that the – usually in practice recommended - thickness of 6-8 mm for the upper mould is not adequate for producing a part with acceptable quality. The reason for choosing a flat plate as a case study is because this is the worst scenario. Parts with curvature will show different behavior than those which are flat. However, the same methodology can be applied and the deformation of the mould can be predicted a priori. For the numerical simulations the experimental data for the perform *Uniconform* from *Saint-Gobain*, for the resin *Polylite 480-500* from *Reichhold* and for the resulting composite have been used. Some of these properties can be found in Table 1.

Property	Uniconform	Polylite 480-500	Composite
Porosity	0.5		
Viscosity		500 mPa·s	
Permeability	$2\cdot10^{-9}$ m^2		
Flexural modulus			6 GPa at 20% weight
Strength			150 MPa at 20% weight

Table 1. Preform, resin and composite properties used in the simulation

*4.1 Case study 1: plate with 1000*1000 mm^2 face area*

The numerical model of the plate used in the parametric analysis is shown in Figure 2. In the left part of Figure 2 the *Ansys CFX* model employed is shown, while in the right one the Ansys Mechanical. In the CFX model the infusion point is at the centre of the plate (denoted with blue arrows) and the outlets are distributed at the periphery of the part (denoted with black arrows). In the *Mechanical* model the four edges of the upper mould are constrained (denoted with A, B, C and D letters) while with red color with indicate the area that withstands the pressure generated inside the mould. The manometric pressure at the mould inlet is 1 bar. For the numerical solution the plate is discretized in finite elements using the built-in mesh generator of ANSYS. The plate is decomposed in 1233 elements with 2721 nodes. A view of the mesh is shown in Figure 3.

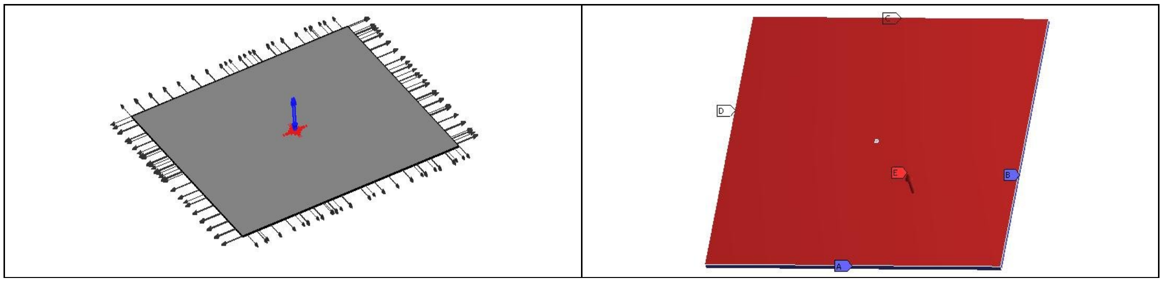

Figure 2. Left Part: *Ansys CFX* computational model **Right Part:** *Ansys Mechanical* computational model

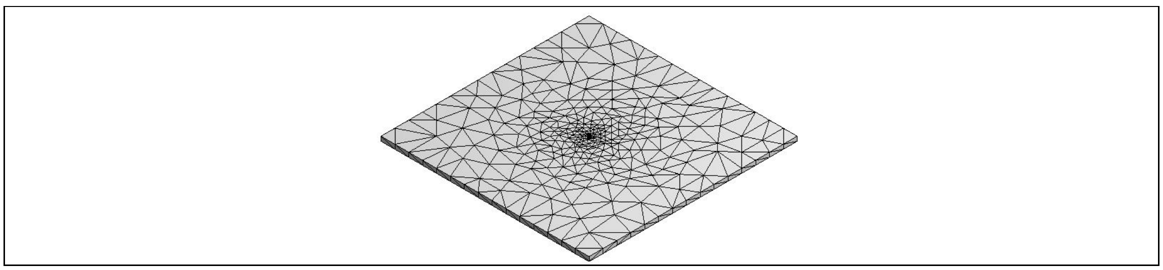

Figure 3. Finite element mesh of the mould

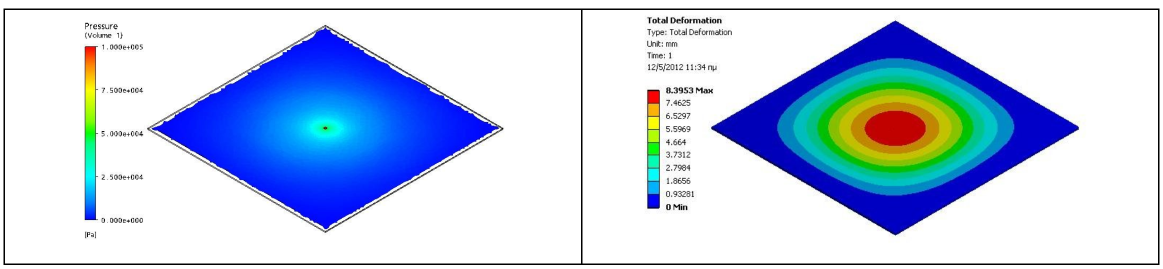

Figure 4. Left part: Pressure field inside the mould cavity **Right part:** Mould deformation

The numerical results for the pressure field inside the mould cavity at the end of the filling process are shown in the left part of Figure 4. In the right part of Figure 4 the resulting deformation of the upper mould for a thickness t=10 mm is shown. In Table 2 the numerical results for different mould thicknesses is shown. In Figure 5 the mould deformation during the mould filling process is shown for a mould thickness t=24 mm. Table 2 summarizes the results of the mould deformation for different mould thicknesses using non-linear analysis (large deformation analysis) respectively. Furthermore, a comparison between a central and a peripheral filling process has been performed. In the peripheral filling process the resin is infused at the periphery and the outlet is located at the center of the part (exactly the opposite with respect to the central filling process). It is interesting to note that in the peripheral filling a much shorter filling time (220 s) has been computed in comparison to the central filling process (450 s). Furthermore, the pressure field during the filling process was more evenly distributed in the peripheral filling. However, at the end of the filling process both strategies create the same pressure field.

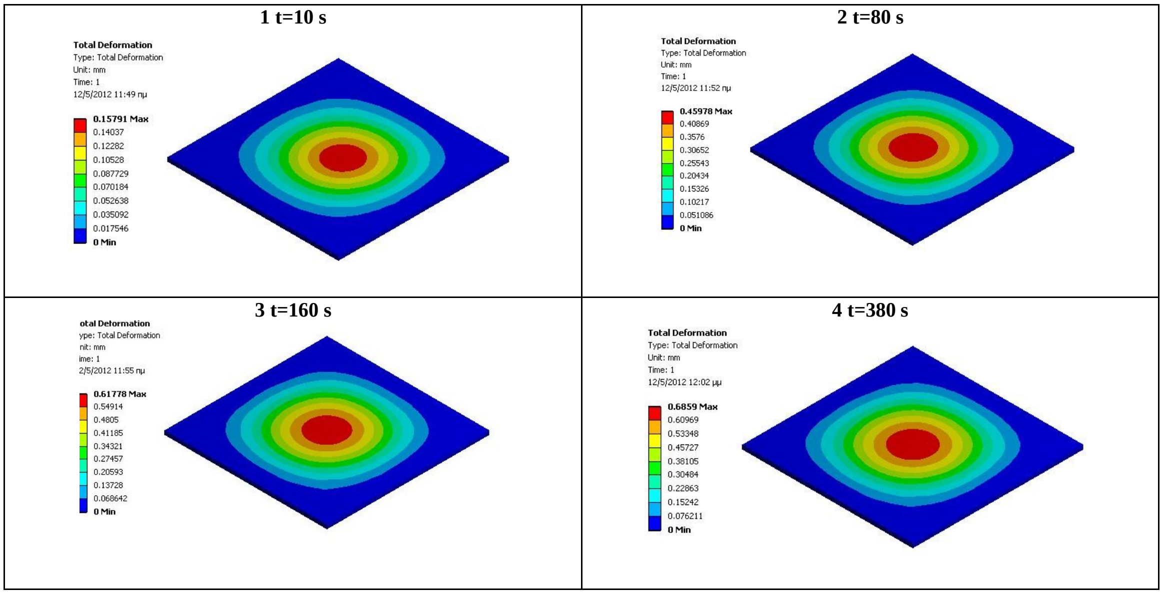

Figure 5. Mould deformation during the filling process

Mould thickness	10 mm	12 mm	15 mm	18 mm	21 mm	24 mm
Maximum Deformation	8.39 mm	5.85 mm	3.30 mm	1.97 mm	1.07 mm	0.73 mm

Table 2. Mould deformation results using large deformation analysis (1000*1000 mm^2)

*4.2 Case study 2: plate with 500*500 mm^2 face area*

The results considering a plate with dimensions *500*500 mm^2* for different mould thicknesses are shown in Table 3. The manometric pressure at the mould inlet is 1 bar.

Mould thickness	5 mm	6 mm	7 mm	8 mm	9 mm
Maximum Deformation	6.16 mm	4.75 mm	3.98 mm	2.56 mm	1.87 mm

Table 3. Mould deformation results using large deformation analysis (500*1000 mm^2)

4 Conclusions

In this study a fluid-structure interaction approach has been proposed for the simulation of mould filling using the Light-RTM technique. Coupling between the mechanical deformation of the flexible cover and the resin flow inside the mould cavity is achieved by implementing an indirect fluid-structure interaction approach using Ansys CFX and Ansys Mechanical software. Based on this model, a general methodology has been developed to simulate numerically the resin flow during the mould filling process. Numerical results have been compared to theoretical as well experimental ones and good agreement was found. According to our knowledge this is the first time such an approach is followed and presented.

A comparison between peripheral and central filling of flat rectangular plates has shown that the first approach is advantageous in terms of filling time and pressure field during filling. However, the resulting pressure field at the end of the filling process is the same for both cases. The numerical results from the structural analysis have shown that the usually recommended thickness for the upper mould can't be used without prior analysis considering the usual expected part quality. A methodology like the one presented in this paper can be used in order to determine the optimum part thickness of location of resin inlets.

Acknowledgments

The results presented in this paper are part the work performed for the research project "Optimization of Composite Production using Resin Infusion" which is co-funded through the Research Promotion Foundation by the Republic of Cyprus and the European Regional Development fund of the EU.

References

[1] Mallick P. K. *Fiber-Reinforced Composites: Materials, Manufacturing, and Design*. CRC Press (2007).

[2] Barbero E. *Introduction to Composite Materials Design*. CRC Press (1998).

[3] Park C., Lee W., Han W.S. , Vautrin A., Simultaneous optimization of composite structures considering mechanical performance and manufacturing cost. *Composite Structures*, **Volume 65**, pp. 117-127 (2004).

[4] Mouton S., Teissandier D., Sébastian P., Nadeau J.P., Manufacturing requirements in design: The RTM process in aeronautics, *Composites: Part A*, **Volume 41**, pp. 125–130 (2010).

[5] Maclaren O., Gan J.M., Hickey C.M.D., Bickerton S., Kelly P.A. *The RTM-Light manufacturing process: Experimentation and modeling* in "Proceeding of *ICCM-17 Conference*, Edinburgh, UK, (2009).

[6] Duan Y., Tan Z., Zhao Y., Sun J., Compression Responses of Preform in Vacuum Infusion Process. *Chinese Journal of Aeronautics*, **Volume 21**, pp. 370-377 (2008).

[7] Rodriguez E., Giacomelli F., Vazquez A. Permeability-Porosity Relationship in RTM for Different Fiberglass and Natural Reinforcements. *Journal of Composite Materials*. **Volume 38**, pp. 259-268 (2004).

[8] Golestanian H. Physical Determination of Permeability Variation with Porosity for Composite Preforms, *International Journal of Engineering Science*, **Volume 18**, pp. 67-73 (2007).

[9] Moaveni S., Finite Element Analysis Theory and Application with ANSYS, Prentice Hall (2007).

[10] Raw M.J., *Robustness of coupled algebraic multigrid for the Navier–Stokes equations*, in: Proceedings of the 34th Aerospace and Sciences Meeting & Exhibit, AIAA 96-0297, (1996).

[11] Barth T.J., Jesperson D.C., The design and application of upwind schemes on unstructured meshes, AIAA Paper 89-0366, 1989.

[12] Raw M.J., Robustness of coupled algebraic multigrid for the Navier–Stokes equations, in: Proceedings of the 34th Aerospace and Sciences Meeting & Exhibit, AIAA 96-0297, 1996.

[13] Rhie C.M., Chow W.L., A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation, AIAA Paper 82-0998, 1982.