Euromech Colloquium 602 – **Composite manufacturing processes. Analyses, modelling and simulations**
March 13-15, 2019, Lyon, France

CUSTOM OPENFOAM SOLVER FOR MODELLING LIQUID COMPOSITE MOULDING PROCESSES

Rohit G. Sebastian[1], Christof Obertscheider[2], Ewald Fauster[3] and Ralf Schledjewski[4]

[1] Christian Doppler Laboratory (CDL) for High Efficient Composite Processing, Montanuniversität Leoben, Otto Glöckel-Straße 2, 8700 Leoben, Austria, rohit.sebastian@unileoben.ac.at
[2] Department of Aerospace Engineering, Fachhochschule Wiener Neustadt, Johannes-Gutenberg-Straße 3, 2700 Wiener Neustadt, Austria, christof.obertscheider@fhwn.ac.at
[3] Processing of Composites Group, Montanuniversität Leoben, Otto Glöckel-Straße 2, 8700 Leoben, Austria, ralf.schledjewski@unileoben.ac.at
[4] Christian Doppler Laboratory (CDL) for High Efficient Composite Processing, Montanuniversität Leoben, Otto Glöckel-Straße 2, 8700 Leoben, Austria, ralf.schledjewski@unileoben.ac.at

Key Words: *process modelling, conservation of energy, computational fluid dynamics, porous media, multi-phase flow, liquid composite moulding, resin transfer moulding.*

Composite materials are increasingly used for the production of lightweight components because of the higher specific strength that they offer in comparison to traditional engineering materials. Because of this, there has been a lot of interest and effort in improving and optimising composite manufacturing processes for the production of higher quality components for lower costs. Liquid Composite Moulding (LCM) is a popular composite manufacturing process in which dry preforms are impregnated with matrix material. The filling efficiency of LCM processes can be improved by using Computational Fluid Dynamics (CFD) to perform filling simulations of the moulds during the mould and process design stage.

Several software tools are capable of running flow simulations. Most purpose-built tools such as PAM-RTM and LIMS usually model the flow around Darcy's law, while most general-purpose tools such as Abaqus/CFD and ANSYS FLUENT are based on the Navier-Stokes equations. For simulating LCM processes, the solver should be capable of solving incompressible multiphase non-isothermal flows through porous media and cure kinetics. However, none of the aforementioned commercially available software packages have all the features required to fully model LCM processes, in addition to which their source code is not available to the public. This makes it impossible for users to understand how the numerical schemes are implemented or for any new features to be implemented without involvement of the original software publishers.

This paper documents the efforts to develop a tool capable of fully simulating LCM processes as an open source alternative to the commercially available software packages. The approach chosen for this task was to extend the functionality of OpenFOAM, a free and open source C++ toolkit built primarily for problems in continuum mechanics. The default OpenFOAM installation includes several solvers, libraries and utilities which are together capable of modelling a wide range of fluid flows.

The OpenFOAM solver that comes closest to meeting the aforementioned requirements for simulating an LCM process is *interFoam*, which is a solver for two-phase incompressible isothermal flow [1]. *interFoam* is used as the starting point for the custom-built LCM simulation tool. OpenFOAM supports dynamic extension of its solvers using the *fvOptions* framework, which includes support modelling of porous media through the *fvOption*, *explicitPorositySource*. The heat transfer between the different phases and also between the preform and the mould is modelled by solving an additional energy balance equation. Since the conventional energy balance equation is derived for open channel flows, the implemented energy balance has to be specially suited for flow through porous media. One of the ways of deriving balance equations for flows through porous media is to average the standard equations over volumes containing several pores [2, 3]. Such a volume-averaged energy balance derived keeping LCM processes in mind [4] is selected for this application. Since the fluid viscosity changes as a function of temperature, a custom viscosity model is also implemented. This gives us a solver with all the aforementioned qualities with the exception of being able to model cure kinetics.

The implementation of a porous medium is validated by comparing simulation results with results as predicted by Darcy's law, which is generally considered to model flow through porous media with reasonable accuracy. For validating the implementation of the new volume averaged energy equation, the results obtained from simulations are compared with RTM experiments in which thermocouples have been inserted into the mould. The primary point of comparison will be the temperature fields predicted by the solver as these can be directly compared with the readings from the thermocouples.

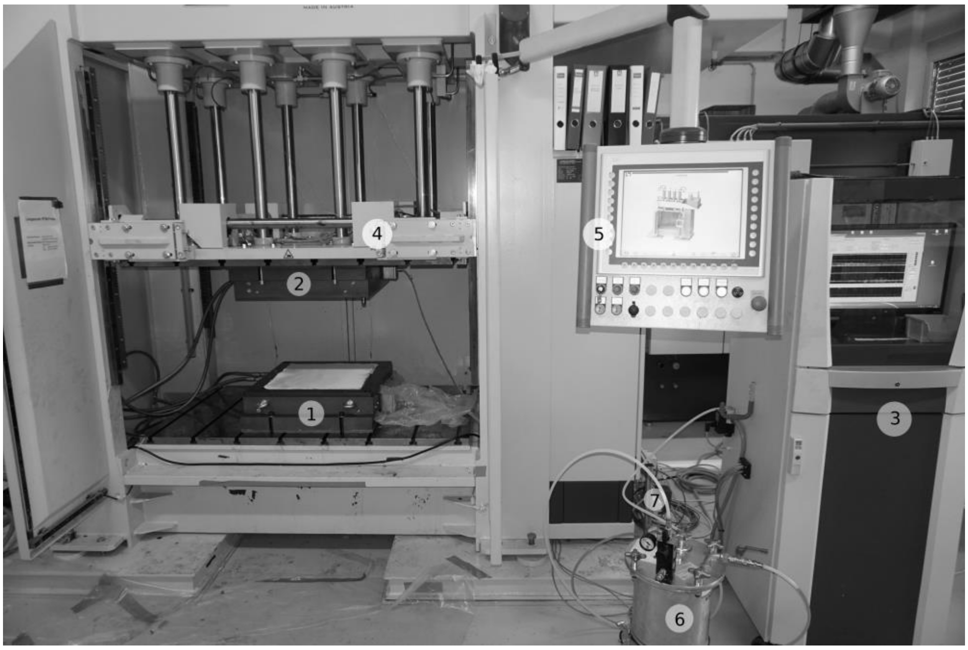

Figure 1 - The experimental setup. 1 - lower half of permeameter; 2 - upper half of permeameter; 3 - control panel for permeameter; 4 - hydraulic press; 5 - control panel for press; 6 - pressure pot; 7 - data acquisition system

REFERENCES

[1] Santiago Márquez Damián. *Description and utilization of interFoam multiphase solver*.

[2] Charles L. Tucker and Richard B. Dessenberger. Governing Equations for Flow and Heat Transfer in Stationary Fiber Beds. In: *Flow and rheology in polymer composites manufacturing*. Advani, S.G., Ed.; Elsevier: Amsterdam, New York, 1994.

[3] D.A. Nield and A. Bejan. *Convection in Porous Media;* Springer New York: Dordrecht, 2013.

[4] Rohit George Sebastian. Extension of OpenFOAM solvers for improved flow front prediction during Liquid Composite Moulding - Rohit George Sebastian. Master's dissertation, Fachhochschule Wiener Neustadt, 2018.