COMPUTATIONAL FLUID DYNAMICS APPLIED TO COMPOSITES MANUFACTURING

T.S. Lundström[1, *]

[1] *Division of Fluid and Experimental Mechanics, Luleå University of Technology, 97187 Luleå, Sweden.*

*Corresponding author (staffan.lundstrom@ltu.se)

Keywords: *Computational fluid dynamics, modelling, vacuum infusion, sheet moulding compound, the autoclave process, twin screw extrusion*

Introduction

Computational fluid dynamics (CFD) has been used to simulate the flow during vacuum infusion [1-2], sheet moulding compound (SMC) [3-4], the auto-clave process [5] and twin-screw extrusion. Three-dimensional geometries were studied and porous media and multiphase flows were treated with a homogenous approach. The advantages and drawbacks of such a methodology are here discussed based on previous work [1-5] and some new results on twin-screw extrusion. To exemplify: For the vacuum infusion process the porous media can be located in a complex mould with moving boundaries and can have an anisotropic, spatial and time dependent permeability. The pressure within the mould during compression moulding of SMC can be predicted if shear-thinning effects are accounted for.

Vacuum infusion

In vacuum infusion the impregnation is characterised by a full 3D flow in a thin porous medium having an anisotropic, spatial- and time-dependent permeability. Also the shape of the porous media alter as a function of time since it is subjected to forces from the difference between the atmospheric pressure and the pressure inside the mould and since the compliance of the fibre network may change as it is wetted, see experimental results in [6-7]. In [1] it was shown how this can be implemented in a general computational fluid dynamic software through written subroutines that couple the equations of the flow to the equations describing the stiffness of the fibre reinforcement, modify the momentum equations to account for the Darcy flow and remesh the computational domain in each time step to account for the deformation caused by pressure changes. The method was verified with a mesh refinement study and analytical solutions and validated with experiments. The strength of the approach was also demonstrated by simulations of mould filling during vacuum infusion of a circumferential stiffener [2]. In addition to the overall development of the model, a couple of issues were studied. Local lead of the flow front due to poor fitting was shown to have a major influence on the filling pattern while effect of lubrication of the fibers and the corresponding ditch trailing the flow front on the fill time were negligible for the case studied. Regardless of which phenomenon in focus it was clear that experimental observations, characterisations and measurements are of outmost importance for the simulations.

Sheet mould moulding compound

Compression moulding of sheet moulding compound has been studied with CFD by applying three different approaches. This includes a surrogate based optimization methodology applied on simulations with a relatively simple model for the rheology [3], a study where different rheological models are compared including shear thinning effect and a study where the layers of the compound nearest to the mould surfaces are given a much lower viscosity than the viscosity in the bulk layers [4]. The pressure can be accurately predicted with both of the latter approaches while with the first method the temperature dependent Newtonian viscosity model was not good enough to describe all details of the flow of the compound. The multi-objective surrogate-based inverse modeling method in it-self worked very well. The shape of the derived flow fronts in [3] was also similar to those obtained in experiments in [8]. An additional result in [4] was that altering the heating time until the prepreg was assumed to start to flow had a significant effect regardless of the considered process settings. Also in this case it is of outmost importance that the material and processing conditions are known.

Autoclave processing

Simulations of heat transfer to a tool placed within an autoclave have been carried out with CFD with and without taking thermal radiation into account [5]. Hot air is forced to flow over the tool and the pressure is increased within the autoclave. The predicted temperature at several discrete positions on the tool is in agreement with experimental measurements presented in [9], especially for simulations when thermal radiation was included in the model. The points with good agreement are often located at positions with relatively simple flow. For some points, however, the agreement was not as good, and these points are often located at positions with more complex flow. The discrepancy may be traced to issues such as the modelling of turbulence and the quality of the mesh for a more complicated flow. Still it may be stated that CFD can be used to predict the heat transfer within an autoclave during manufacturing of composites and thermal radiation should be considered when modelling heat transfer for such a case. The study also shows that it is important to use a proper inlet velocity profile. Simulation results with a height-dependent velocity profile compared much better to the experimental results than those obtained when the average velocity inside the autoclave was set as the inlet velocity profile. Even better agreement may be achieved by disclosing the full velocity profile inside the autoclave. A final result was that small differences in tool symmetry resulted in rather large temperatures differences for the process setting applied in the simulations. This behavior was also observed in the experimental study in [9].

Twin-screw extrusion

Liquid-mediated melt compounding of nanocomposites in twin-screw extruders is an emerging method that combines solution-assisted and melt mixing methods in such a way that the disadvantages of these two methods is overcome. The liquids used work both as temporary carrier of nanomaterials as plasticizer after evaporation of the liquid. Modelling the flow of thermoplastic polymers in twin-screw extruders is challenging and the mixing, the changing of volumes of an evaporating solvent and the non-spherical particles add considerably to the complexity. In the current case a two-phase flow approach is chosen and the particles are assumed to follow the carrier fluid through the mixing section of the extruder. The mixing elements are modelled with the immersed boundary method and results are presented in the form of mixing ratio and areas of maximum shear.

Acknowledgements

The author acknowledge the authors to the papers [1-5] below: Magnus Andersson, Rikard Gebart, Nils Langhans, Daniel Marjavaara, Sofia Ebermark, Jimmy Kluge, Lars-Göran Westerberg, Kurt Olofsson and Tony Nyman. The author also acknowledge Gustaf Alnersson for doing the simulations and analysis on Extrusion and Yvonne Aitomäki and Kristiina Oksman for discussions on the same subject.

References

[1] Andersson, H.M., Lundström, T.S., Gebart, B.R., "Numerical model for vacuum infusion manufacturing of polymer composites", Numerical Methods for Heat and Fluid Flow, 13, 2-3, pp. 383-394 (2003)

[2] Andersson, H.M., Lundström, T.S., Langhans, N. "Computational fluid dynamics applied to the vacuum infusion process" Polymer Composites, 26, April, pp. 231-239 (2005)

[3] Marjavaara, B.D., Ebermark, S., Lundström, T.S., "Compression Moulding Simulations of SMC using a Multi-objective Surrogate-based Inverse Modeling Approach" Mechanics of Composite Materials 45 (5), 739-758 (2009)

[4] Kluge, NEJ, Lundström, TS, Westerberg, LG, Olofsson, K, "Compression Moulding of SMC: Modelling With CFD and Validation" Journal of Reinforced Plastics 34: 479-492, (2015)

[5] Kluge, N.E.J, Lundström, T.S., Westerberg, L.G., Nyman, T., "Modelling Heat Transfer Inside an Autoclave: Effect of Radiation", Journal of Reinforced Plastics, Published on-line (2016)

[6] Andersson, H.M., Lundström, T.S., Gebart, B.R., Långström, R., "Flow enhancing layers in the vacuum infusion process", Polymer Composites, 23, 5, pp. 895-901 (2002)

[7] Andersson, H.M., Lundström, T.S., Gebart, B.R. and Synnergren, P., "Application of digital speckle photography to measure thickness variations in the vacuum infusion process", Polymer Composites, 24, 3, (2003)

[8] Odenberger, P.T., Andersson, H.M., Lundström, T.S., "Experimental flow-front visualisation in compression moulding of SMC" Composites Part A, 35, pp. 1125-1134 (2004)

[9] Kluge, N.E.J, Lundström, T.S., Ljung, A-L, Westerberg, L.G., Nyman, T., "An Experimental Study of Temperature Distribution in an Autoclave", Journal of Reinforced Plastics, Published on-line (2016)

Computational fluid dynamics was born principally in the aerospace field as a method for fluid flow and heat transfer research methods following experimental and analytical approaches. Along with progress in the cost performance of computers, computational fluid dynamics is now establishing itself as a tool to improve production processes and product quality in the steel, nonferrous metals, glass, plastics, and composite materials industries. Materials manufacturers use computational fluid dynamics for diverse purposes: 1. Reduction in experimental conditions and costs; 2. Detailed analysis of mechanisms with multifaceted information unobtainable through experimentation; 3. Universal tool for scale-up; and 4. Evaluation of novel processes. An Introduction to Composite Materials Composite Manufacturing â€“ Autoclave Variability. Composites Manufacturing. On July 12, 2012 Â· In Composite Materials, Manufacturing.Â In terms of manufacturing advanced fibre-reinforced composites the single most important aspect to recognize is that the material and the structure are created at the same time. Consequently any defects that are induced during the manufacturing process directly influence the strength and stiffness of the material and structure.Â Computational Fluid Dynamics (2). Contra-Rotation (1). Design (2). Lecture 1 - Introduction to CFD Applied Computational Fluid Dynamics. AndrÃ© Bakker. 12. Fluid dynamics. 1. Introduction to CFD. â€¢ Fluid dynamics is the science of fluid motion. â€¢ Fluid flow is commonly studied in one of three ways: â€“ Experimental fluid dynamics. â€“ Theoretical fluid dynamics. â€“ Numerically: computational fluid dynamics (CFD). â€¢ During this course we will focus on obtaining the knowledge required to be able to solve practical fluid flow problems using CFD. â€¢ Topics covered today include: â€“ A brief review of the history of fluid dynamics.Â manufacturing. â€“ Heat transfer for electronics packaging applications. â€“ And many, many more! Computational fluid dynamics. From Wikiversity. Jump to navigation Jump to search. Part of the Wikiversity Division of Fluid Mechanics, Division of Applied Mechanics, School of Engineering and the Engineering and Technology Portal. CFD Simulation of the fairing aerodynamics of an Ã©co-mobile (velomobile). Contents.Â 1.1 Computational Fluid Dynamics(CFD) Applications. 2 Learning Materials. 2.1 Texts. Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved.Â Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions.