VERIFICATION OF FLOWTEX SOLVER USING ANSYS CFX - EXAMINING THE PERMEABILITY PREDICTION METHOD ON A RANGE OF TEXTILE ARCHITECTURE MODELS

Elinor E. Swery[1*], Reinhold Meier[2], Stepan V. Lomov[3], Christoph Hahn[2], Piaras Kelly[4], Ilya Straumit[3]

[1]*Centre for Advanced Composite Materials, Department of Mechanical Engineering, University of Auckland, Private Bag 92019, Auckland, New Zealand*
[2]*Institute for Carbon Composites, Technische Universität München, Faculty of Mechanical Engineering, Boltzmannstraße 15, D-85748 Garching b. München, Germany*
[3]*Department of Metallurgy and Material Engineering, Katholieke Universiteit Leuven, Leuven, Belgium*
[4]*Department of Engineering Science, University of Auckland, Private Bag 92019, Auckland, New Zealand*
[*]*corresponding author: elinor.swery@auckland.ac.nz*

Keywords: Permeability, Textile Modelling, Computational Simulation.

Abstract
Predicting reinforcement textile permeability properties through simulation approaches is a repeatable and robust method which can capture complex behaviours during Liquid Composite Moulding processes. This paper compares the predicted permeability values obtained from conducting flow simulations on the same textile models using the FlowTex and Ansys CFX solvers. An automated tool, which has been developed and is presented, has been used for this, performing flow simulations on WiseTex generated meshes using Ansys CFX. This tool converts the voxel files exported from WiseTex into voxel meshes compatible with Ansys, representing the volume of resin within the unit cell. The voxel meshes are automatically cleaned, deleting any floating elements and the boundary regions are defined based on the unit cell size. The permeability comparison results for a range of idealised textile models are presented. Furthermore, the ability of both solvers to capture variations in unit cell geometric parameters is demonstrated. Finally, both solvers were used to predict the permeability values of unit cell models based on the textile used in the second permeability benchmark exercise [1]and these are compared with the experimental permeability values.

1. Introduction

Fibre Reinforced Polymer Composite (FRPC) materials are used in a large number of industrial applications. FRPCs consist of two or more distinct materials (generally fibre reinforcement and polymer matrix), forming a material with more desirable properties. For applications where parts are mass produced and high levels of accuracy and repeatability are required, Liquid Composite Moulding (LCM) processes are the preferred manufacturing method [2]. In LCM processes, resin is injected into a mould containing the dry fibrous reinforcement and is left to cure before the final part can be removed from the mould.

The use of LCM simulations as process design tools is increasing in industry. These simulations are used to accurately predict fill time, flow front advancement and dry spot formation, ultimately enabling the production of complex high quality parts under the most efficient conditions [2]. These simulations require knowledge of the reinforcing material's permeability characteristics. Permeability, **K**, is a measure of the ability of a reinforcement material to transmit fluids. It is an important material characteristic that determines the flow propagation of the resin during LCM manufacturing processes and is an indispensable input into LCM process simulations. As the permeability behaviour of reinforcing textiles is a strong function of the textile's complex architecture [2] (which therefore makes the development of adequate analytical models extremely difficult), many researchers are relying on measurements obtained from experiments. For this, many data points are required to capture the influence of varying compaction levels, preform structure, applied shear etc. (as there are no models to accurately capture these effects). This is time and labour intensive as and the results can vary significantly as a result of a lack of a standardised testing method and errors involved in the handling process [3].

Another method used to obtain the permeability characteristics of reinforcing textiles is through simulation approaches. Textile modelling techniques are used to generate models of the reinforcement unit cell. By imposing a pressure drop across a mesh generated from these models and solving the governing flow equations the permeability properties may be determined (using Darcy's law) [4]. A number of methods are currently available that enable the implementation of textile modelling techniques. The most common are the WiseTex suite, developed at KU Leuven [5] and TexGen, developed at the University of Nottingham [4].

To solve the governing flow equations, commercial Computational Fluid Dynamics (CFD) software, such as Ansys CFX may be used, however this has traditionally been viewed as a computationally intensive and an expensive option [4, 5]. Instead, methods such as the Stream Surface and Grid average methods have been used, which simplify the complex 3D flow problem to 2.5D and 2D respectively. Within the WiseTex suite, an additional tool, FlowTex, has been developed, which can be used to solve the Stokes equation for the flow between yarns [5] (as a simplification of the full Navier-Stokes equation as this is valid for flow with low Reynolds number [5]) and thus compute the permeability. The numerical solver for this is based on the Navier-Stokes finite difference solver, NaSt3DGP, developed at the Institute for Numerical Simulation, (University of Bonn) [6] which has been extended with a finite difference Stokes solver [7]. The Ansys CFX solver on the other hand, uses finite elements discretised domain and solves the full 3D numerical analysis based on the continuity equation.

This paper compares the permeability values obtained using the FlowTex and Ansys CFX solvers neglecting the intra-yarn porosity. With sufficiently similar results, it may then be assumed that the FlowTex solver may confidently be used in place of commercial CFD solvers. FlowTex also has an option to include intra-yarn flow in the calculation, using Brinkman equations. This possibility is absent in ANSYS and hence was not explored.

In order to compare the FlowTex and Ansys solvers in a controlled manner, all the solver options, boundary conditions, set up parameters etc. had to be identical. More importantly, it was crucial to ensure that the mesh used in both simulations was exactly the same. The WiseTex software was used to create single layer models of different reinforcement unit cells. This software implements a generalised description of the internal structure of textile reinforcements on the unit cell level based on models developed at K. U. Leuven [8]. FlowTex was used to generate the representative volume element (RVE) voxel meshes based on the WiseTex textile models. The RVEs are discretized with a collocated grid, distinguishing between solid and fluid elements. As the voxel meshes generated within FlowTex were not compatible with Ansys CFX, an automated tool developed for this work was used to convert them to meshes that can be directly used in Ansys CFX.

For both, the FlowTex and Ansys meshes, non-slip wall conditions were imposed on the top and bottom RVE surfaces as well as on the boundaries between the fluid domain and the fibre yarns. Translational periodic boundaries were used for the two sides, to reflect the periodic structure of the textile [5] and a pressure drop was applied across the injection direction. This is shown on a simple plain woven model in Figure 3. The final permeability value for the RVE was then computed using Darcy's law [9]:

$$\mathbf{q} = -\frac{1}{\mu}\mathbf{K}\nabla \mathrm{P}\,. \tag{1}$$

In order to compute the permeabilities of the RVE in different directions (e.g. weft and warp), the location of the boundaries were altered, but the same mesh used. It is important to note, that the FlowTex solver uses dimensionless variables to compute the permeability, therefore eliminating the necessity to specify parameters such as resin viscosity and injection pressure. Contrary to this, the parameter values used in Ansys represent typical manufacturing cases.

2. Automated conversion and simulation execution tool

A fully automated tool has been developed, which converts the voxel files produced in FlowTex, executes the flow simulations within Ansys CFX and computes the permeability from the simulation result. This tool performs the steps outlined in Figure 1 directly from the Matlab environment without any user intervention. When a step is completed using a different software package, the appropriate script is generated and then executed from within Matlab. This automated method enables the efficient analysis of a large number of voxel files. Outputs from these steps are shown in Figure 2.

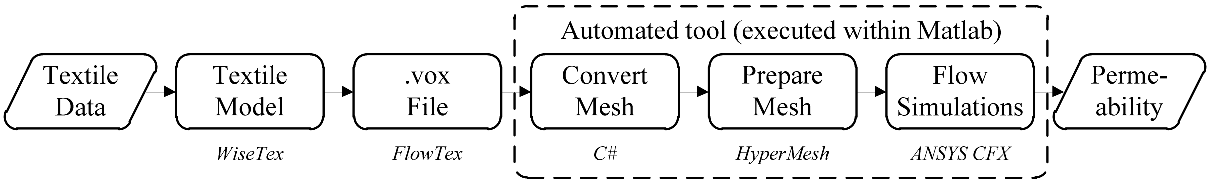

Figure 1. Automated tool steps

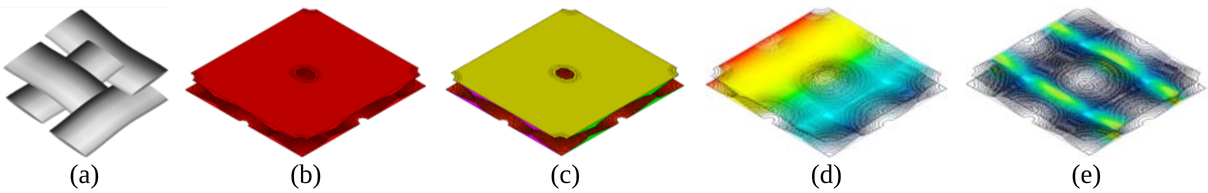

Figure 2. Automated tool steps. (a) Plain Weave WiseTex textile model, (b) converted voxel-mesh, (c) prepared voxel-mesh, (d) resulting pressure contour and (e) resulting velocity contour

The voxel files exported from FlowTex contain all the information required to produce an identical mesh compatible with Ansys CFX simulations. As detailed in [5], information is provided about the regular grid used to create the mesh. The number of nodes used for the regular grid, as well as the spacing of the nodes are given. The voxel file also contains a description of the domain to which each voxel element belongs to. Either the *Solid* domain, (where a tow is found) or the *Fluid* domain (where there is no tow; hence resin can flow).

Two matrices used to describe the resulting mesh are produced using a C# code compiled in Microsoft Visual Studio and executed through Matlab. The first, describing the nodes, contains the node ID numbers, x, y and z locations. The second matrix, describing the voxel elements, contains the element ID numbers along with the eight node IDs required to define that voxel. Since the simulations in this work are only concerned with inter-yarn flow, it is assumed that intra-yarn porosity is zero and that no fluid will flow through voxels in the solid domain. As such, any voxel elements within the solid domain are deleted and not included in the exported element matrix.

These matrices are used to generate a mesh file which is then imported into HyperMesh. Here, the mesh is prepared for the Ansys simulations by automatically deleting any floating voxels (defined as voxels that are not connected to the main mesh) in order to enable the efficient execution of the flow simulations. The boundary regions are defined (as described in Figure 3), where the appropriate elements for each region are located by intersecting 2D planes at the corresponding locations to the regions corresponding 2D planes with the mesh. Lastly, the mesh is exported from HyperMesh as a .cas file, with the appropriate collectors representing the boundary regions, ensuring that it is ready to be used in Ansys. These HyperMesh steps are all completed automatically from within Matlab, through the use of Tool Command Language (TCL) scripts, developed for specifically for this purpose.

The flow simulations are implemented in Ansys CFX through the use of python scripts executed from Matlab. The resulting mass flow rate, $\dot{m}$, was used to compute the unit cell meso-scale permeability. This is illustrated in Equation 2, a modified version of Darcy's law, where L is the unit cell length across which the pressure drop is applied, A, the cross section area, μ, the fluid (resin) viscosity and ρ the fluid density.

$$\mathbf{K} = \frac{\dot{m}\mu L}{A\rho\nabla P} \tag{2}$$

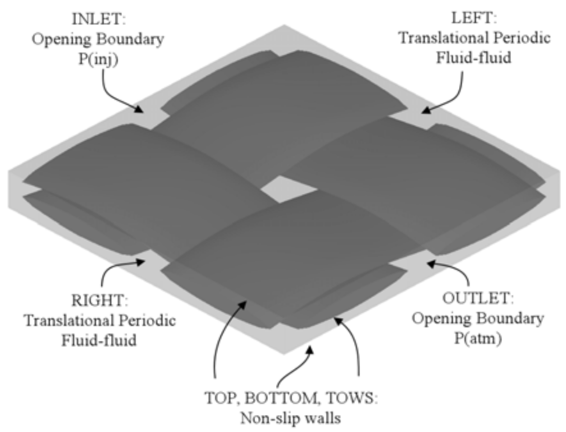

Figure 3. RVE Boundary Conditions

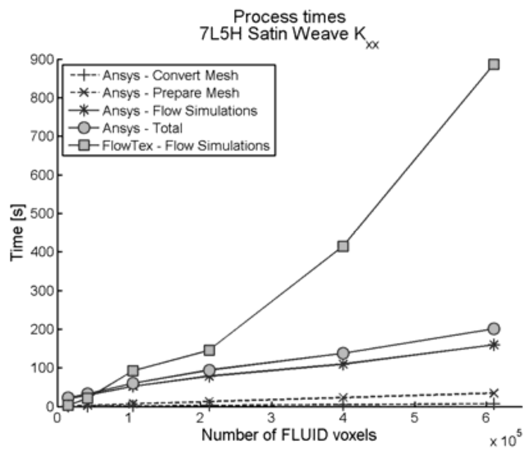

Figure 4. Process times for a Satin Weave

Figure 4 presents the associated processing times to complete each step on Satin Weave meshes with different numbers of voxels. These times were achieved by executing the process on a Windows 7, 64 bit computer, with 8GB RAM and an Intel® Core™ i5-2500 CPU at 3.30GHz. As can be seen, even for the largest mesh comprising of 6.1×10^5 fluid elements (3.0×10^6 total elements), the complete process (using the Ansys solver) was relatively fast, taking a total of 200s. The longest component of the process, conducting the flow simulations, took 159s, and converting and preparing the mesh 7s and 34s respectively. The FlowTex solver took 887s to compute the permeability for the same mesh. The Ansys solver was faster due to the fact that this commercial software has been greatly optimised.

3. Permeability comparison of different geometries

Five different textile geometries were generated in WiseTex using the standard library available in the software; plain weave, ±45° biaxial non crimped non stitched unidirectional preform (UDP), 2D braided, 7L5H satin weave and 2×2 twill weave (based on the material used in the recent permeability benchmark exercise [1]). Single layer unit cells models (representative repeating units of the textile geometry) were generated for each (Figure 5).

(a) (b) (c) (d) (e)

Figure 5. Geometries explored; (a) Biaxial UDP, (b) 2D Braid, (c) Plain weave, (d) Satin weave (e) Twill weave

For each of the textile models, a range of discretisation levels was used to generate voxel files of varying refinements. As these textile geometries have features of similar length scales in the x and y directions, the voxel element dimensions in these directions, dx and dy, were kept equal. The features in the z direction are significantly smaller; hence it was important to ensure that dz was sufficiently small to capture the level of detail required. In order to reflect this, as well as to avoid complications due to high element aspect ratio, the following ratio of voxel dimensions was maintained [10];

$$dx = dy = 10\,dz \tag{3}$$

The K_{xx} and K_{yy} permeabilities, corresponding to warp and weft directions respectively, were computed using the FlowTex and Ansys CFX solvers as described previously and the values compared. These results were obtained with the FlowTex solver precision value (residue to which the system of equations are solved) set to 1×10^{-4} and the maximum number of iterations was set to 1×10^{4}, however none of the simulations executed reached this value. It is important to note that the maximum mesh size that was able to be used was restricted by the FlowTex solver. Generally, voxel meshes containing more than 1×10^{6} *fluid* elements could not be used in FlowTex; these larger meshes introduced problems associated with memory use within FlowTex.

The converged results are presented in Table 1, where the permeability values as well as the anisotropy ratio are given for the finest mesh modelled for each of the geometry models. It can be seen that there is a clear agreement between the two solvers. The K_{xx} and K_{yy} permeability values using the two methods are within a close range, capturing the effects of the various geometries in a similar manner.

Textile	No. of Voxels			K_{xx} [m^2]		K_{yy} [m^2]		K_{xx}/K_{yy}	
Geometry	x	y	z	FlowTex	Ansys	FlowTex	Ansys	FlowTex	Ansys
Biaxial UDP	87	87	135	7.38E-11	7.02E-11	7.38E-11	6.97E-11	1.00	1.01
2D Braid	103	103	48	2.15E-9	2.14E-9	2.15E-9	2.14E-9	1.00	1.00
Plain Weave	109	109	89	2.12E-10	2.07E-10	2.49E-10	2.45E-10	0.85	0.85
7L 5H Satin	197	197	77	3.42E-11	3.68E-11	3.21E-11	3.52E-11	1.07	1.05
Twill Weave	148	150	45	2.05E-10	2.00E-10	2.38E-10	2.36E-10	0.86	0.85

Table 1. Table summarising results for converged values for each of the geometries

4. Permeability benchmark exercise fabric - 2×2 Twill Weave

A permeability benchmark exercise, aiming to obtain comparable results of the in-plane permeability values of a carbon fabric was carried out as detailed in [1]. Using textile models based on the fabric tested there (400 tex G0986 D1200 Carbon 2×2 Hexcel twill weave fabric), enables the comparison of the predicted permeability values of the two solvers with verified experimental results. Images of this textile were obtained by scanning a single layer of it placed in a glass mould, compacted to a set thickness as detailed in [7]. These images were analysed using the image analysis method previously described in [11]. A section of an analysed image is shown in Figure 6, where paths of the tow edges located. The resulting geometric parameters from this analysis are presented in Figure 7 and in Table 2 (along with the values from the supplier's data sheet and measured values that were previously performed according to ISO 10120:1991 and ISO 3801:1977 in [1]). The slight differences between the measured and the image analysis values obtained arise from the fact that the measured values were of uncompacted textile as well as the inevitable material variations present. The theoretical areal weight value of the WiseTex model is also within a realistic range.

Data source	**Tow Width [mm]**		**Construction [tows/cm]**		ρ_A **[g/m²]**	
	Warp	Weft	Warp	Weft	Measured	Model
Data sheet	n/a	n/a	3.5	3.5	285	n/a
Measured	2.31 ± 0.17	2.27 ± 0.20	3.52 ± 0.07	3.46 ± 0.07	284 ± 2	279.5
Image Analysis	2.44 ± 0.23	2.28 ± 0.22	3.45 ± 0.10	3.48 ± 0.11	n/a	277.9

Table 2. Reinforcement architecture data

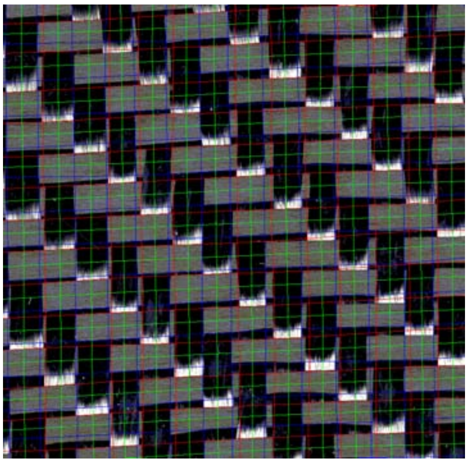

Figure 6. Textile image analysed

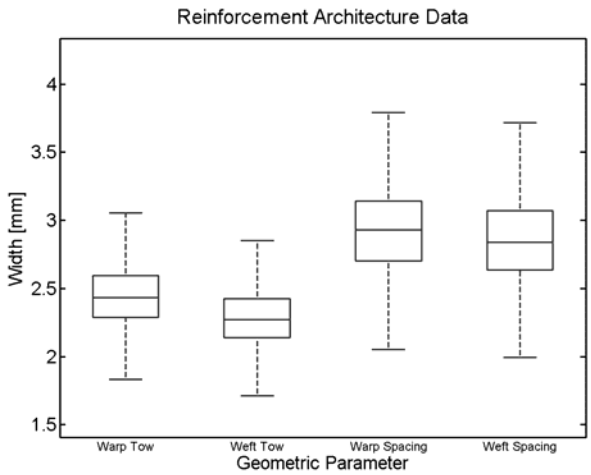

Figure 7. Measured geometric parameters

4.1 Varying geometric parameters

Focus was placed on varying the specific geometric parameters of the Twill Woven model to assess whether both solvers capture the influence of these variations in the same manner. The tow widths and gap sizes for both the warp and weft yarns were altered independently, assessing the mean value and 1 and 2 standard deviations smaller and larger than the mean, while maintaining the remaining parameters at their mean values. The tow thickness was maintained at 0.175mm for all tests, equivalent to the compaction level during the scanning, resulting in a 45% fibre volume fraction. The models were created in WiseTex, and the meshes created based on these, using the largest level of discretisation that was able to be solved in FlowTex. The resulting predicted permeability values are presented in Figure 8. The xx and yy directions were defined respectively in the warp and weft of the material.

It is evident from these plots that both solvers capture these effects on permeability in the same manner and as would be expected; increasing the weft tow width significantly reduces the weft permeability values while only slightly decreasing the warp permeabilities. Similarly, increasing the weft tow spacing, significantly increases the weft permeability and increases the warp permeability to a lesser extent. The predicted permeability values in the two directions are within a close range confirming that both solvers may confidently be used.

4.2 Comparison with experimental values

In the second permeability benchmark exercise [1], mean values of $0.80 \pm 0.15 \times 10^{-10} m^2$ at a fibre volume content of 44.98 $\pm$ 1.1 % and $1.31 \pm 0.26 \times 10^{-10}$ m^2 at a fibre volume content of 44.8 $\pm$ 2.0 % were obtained for the warp and weft permeabilities respectively (see dashed straight lines in Figure 8). These values are lower than the predicted permeability values obtained using the two solver types. It is promising however that the predicted values are within the same order of magnitude as the experimental values.

The most probable explanation for the difference is the fact that only single layer models were used, whereas the experiments in the benchmark exercise were conducted on stacks consisting of 10 layers. Using single layer models to predict the permeability does not capture the effect that multi-layer nesting may have on the permeability; yarns from adjacent layers nesting into the gaps thereby restricting the fluid flow and decreasing the permeability.

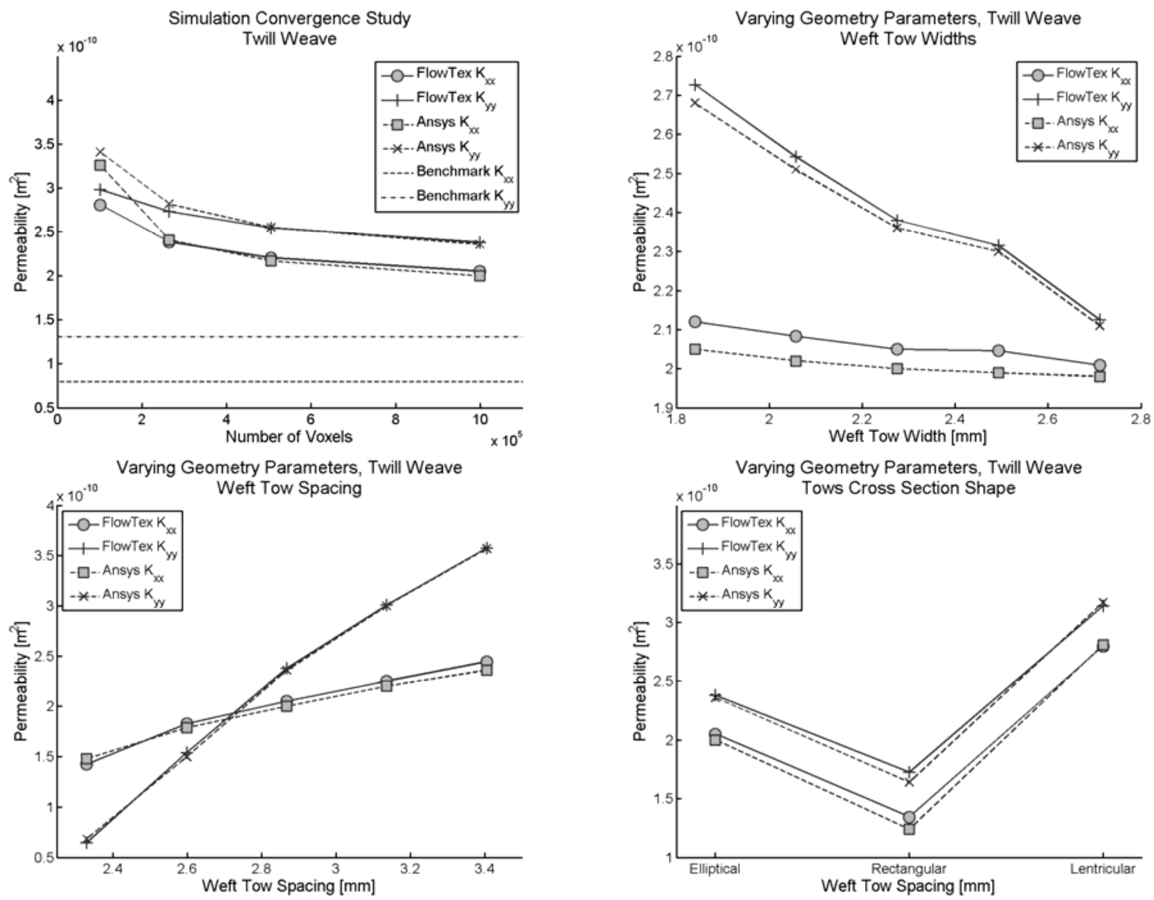

Figure 8. Effects on permeability by varying textile geometry compared to experimental results

Conclusions

This work shows that for the same meshes, the FlowTex solver predicted similar permeability values as Ansys CFX. This means that FlowTex may confidently be used, avoiding high costs associated with using a full commercial CFD program. The use of FlowTex also eliminates the need for expert knowledge in order to set up and execute complicated simulations. It was, however, also illustrated that the computation times of Ansys CFX are significantly lower, which may be relevant for simulations involving large meshes. Further, the mesh size was restricted in FlowTex due to problems associated with memory usage within FlowTex.

The conversion tool presented in this study may be used to convert meshes produced from WiseTex models in order to compute their permeabilities in Ansys CFX, providing an easy to use link between the two software. This also enables the prediction of permeability using large meshes as well as for more complex simulations, for example involving the effects of non-Newtonian fluids or resin curing reactions combined with the flow models.

The predicted permeability values obtained using both solvers are higher than the experimentally obtained values. This may be attributed to the fact that only single layer models were used, thus not capturing the effects of multi-layer nesting. This will be further explored.

Acknowledgements

The authors would like to acknowledge the assistance received from Itamar Amith with optimising the C# code, and Bart Verleye for his assistance with FlowTex. The authors also gratefully acknowledge the funding provided by the German Academic Exchange Service (DAAD) which was used in the initial stages of this collaboration work.

References

1. Vernet, N., et al., *Experimental Determination of the Permeability of Engineering Textiles: Benchmark II.* Composites Part A: Applied Science and Manufacturing, 2014, in press.
2. Trochu, F., et al., *Advanced Numerical Simulation of Liquid Composite Molding for Process Analysis and Optimization.* Composites Part A: Applied Science and Manufacturing, 2006. **37**(6): p. 890-902.
3. Arbter, R., et al., *Experimental determination of the permeability of textiles: A benchmark exercise.* Composites Part A: Applied Science and Manufacturing, 2011. **42**(9): p. 1157-1168.
4. Wong, C.C., et al., *Comparisons of novel and efficient approaches for permeability prediction based on the fabric architecture.* Composites Part A: Applied Science and Manufacturing, 2006. **37**(6 SPEC. ISS.): p. 847-857.
5. Verleye, B., *Computation of the Permeability of Multi-Scale Porous Media with Application to Technical Textiles*, 2008, Katholieke Universiteit Leuven: Leuven. p. 212.
6. Griebel, M., T. Dornsheifer, and T. Neunhoeffer, *Numerical simulation in fluid dynamics: a practical introduction*. Vol. 3. 1998: Siam.
7. Lomov, S.V., et al., *Meso-level textile composites simulations: Open data exchange and scripting.* Journal of Composite Materials, 2013.
8. Verpoest, I. and S. Lomov, *Virtual textile composites software : Integration with micro-mechanical, permeability and structural analysis.* Composites Science and Technology, 2005. **65**(15-16): p. 2563-2574.
9. Darcy, H. in *Les Fontaines Publiques de la Ville de Dijon*. 1856. Dalmont, Paris.
10. Felippa, C.A., *FEM Modeling: Mesh, Loads and BCs*, in *Introduction to Finite Element Methods*. 2004, Department of Aerospace Engineering Sciences and Center for Aerospace Structures, University of Colorado: Boulder, Colorado 80309-0429, USA.
11. Swery, E.E., et al., *Automated geometric characterisation of woven reinforcing textiles using image analysis techniques*, in *34th Risø International Symposium on Materials Science*2013: Roskilde, Denmark.