MULTI-SCALE FINITE ELEMENTS ANALYSIS OF BRAIDS AND BUNDLES OF FILAMENTS USING EMBEDED ELEMENTS

B. Burgarella[1], L. Laberge Lebel[1]

[1] LabSCFA, Polytechnique Montréal, Montréal, Canada

* Corresponding author (boris.burgarella@polymtl.ca)

ABSTRACT

Predicting the behavior of dry fibers in different conditions is a key aspect of process modeling of composite materials. However, predicting the behavior of bundles consisting of thousands of individual filaments can be very costly in terms of computer power. A new method multi-scale is proposed that feature embedded beam elements in truss elements. Very small beams are used to model the flexion while truss elements allows to estimate traction and contact responses. This method is compared and benchmarked against estimations given by pure beam and pure truss models and a reference case. The embedded elements seems to provide a better estimation as the cumulative error over all the chosen indicators is much less than pure beams and truss.

KEYWORDS: *Filament bundle, multi-scale, finite elements method*

INTRODUCTION

With the ever-increasing need of reducing carbon emission, many industries are working towards structure weight reduction. Designing new structures using composite materials seems to be part of the solution. Thanks to their very good specific strength and stiffness, composite materials allows designers to reach new levels of structure optimization that were not achievable with classical materials. However, composite materials feature multiple scales which can complexify the computation of their behavior. Usually three main scales are considered. The macroscopic scale, which corresponds to the structure scale (a full plane, building, or even a specific part of it). The mesoscopic scale, corresponding to the fabrics scale. Finally, the microscopic scale where the individual filaments are considered.

Author usually address this separation of scales problem by using homogenization methods. These can take the form of full field homogenization methods (Moulinec and Suquet 1994; Moulinec and Suquet 1998; Feyel 2003). Or mean filed homogenization methods (Mori and Tanaka 1973; Castañeda and Willis 1999; Ghossein and Lévesque 2014). These methods allows the estimation of the effective properties of composite materials in structural problems where the fibers are immersed in a solid polymer matrix. But they are not suited for the modelling and prediction of the behaviour of dry fibers. In the absence of a linking matrix, the behaviour of the composite is very different as the filaments are free to move in independent motions. This specificity of dry fibers induce a need of specific methods to fully capture their behavior. It is for example very hard to model the flexion behaviour of a tow as the second moment of area will be too high to match the flexion capabilities of individual filaments

Dry fibers have been studied a lot in ballistics (Zhou, Sun et al. 2004; Nilakantan, Keefe et al. 2010; Wang, Miao et al. 2010; Nilakantan 2013), Zhou *et al.* (Zhou, Sun et al. 2004) developed the so-called Multi-chain digital element method where they aim at approximating the behaviour of filaments by using assemblies of very small truss elements. This approach addresses the flexion problem by removing it completely as truss nodes cannot transfer flexion moments. Wang *et al.* and Grujicic *et al.* obtained very promising results using this method but their work has also received criticism because of the strong hypotheses that are associated with the use of truss elements (Nilakantan 2013). Other authors used an approach based on beam elements (Döbrich, Gereke et al. 2016). In their approach, the tow is discretized and represented as a bundle of beam elements and a convergence study is conducted. Finally some authors adopt a full scale approach by modelling every individual filaments (Nilakantan 2013). This approach, while probably being the most accurate cannot realistically be applied to most of the composite materials because of the very high number of filaments involved. Indeed, carbon fiber tows usually feature from 3000 to 12000 filaments the number of degrees of freedom needed to fully model such a material would be complicated with today's calculation capacities.

In this work, we propose an approach similar to the one used by Döbrich *et al.*(Döbrich, Gereke et al. 2016) as we want to estimate the behavior of tows with a limited number of elements in the cross section. The novelty is that we propose to used embedded elements by embedding a small beam into a larger truss element. Embedded elements were initially developed by Fish (Fish 1992; Fish, Markolefas et al. 1994) in what he called the s-version of the FEM. By superposing two meshes, he was able to reach a higher convergence rate by avoiding excessively deformed elements in regions where very little space is available. This method was later used with success by Tabatabei *et al.* (Tabatabaei, Lomov et al. 2014) in the context of composite materials and their homogenization.

The first section of this paper is dedicated to the definition of the methodology used to benchmark our approach. Then, in a second section, the embedded beams are explicitly defined, in a third second section, we address the validation of the reference case. Finally, in a fourth section, results obtained by this method are presented and its performance is benchmarked against three models: a full beam, a full truss and a reference case.

1 METHODOLOGY

To benchmark the proposed approach, it was decided to put the braid in tension over a square sectioned analytical solid. The solid corresponding to an ideal mandrel over which the braid would be put before injecting the resin. The solid has a 1.5875 mm side and is infinitely long. Four cases were tested: The reference (beam elements, matching the real filaments geometry) with 100 filaments per tow, a truss elements model with 7 filaments per tow, a beam elements model with 7 filaments per tow, and finally an embedded elements model with 7 filaments per tow. Three parameters are studied to benchmark the elements: The braid thickness, the angle between the tows and the final length of the braid. The braid thickness is an important parameter as it need to be known in order to correctly design the mold that will be used to inject the resin. The angle between the tows has a strong influence on the mechanical properties of the final piece. Finally the final length was chosen as it provides a good estimation of the dry braid stiffness.

2 MODEL DEFINITION

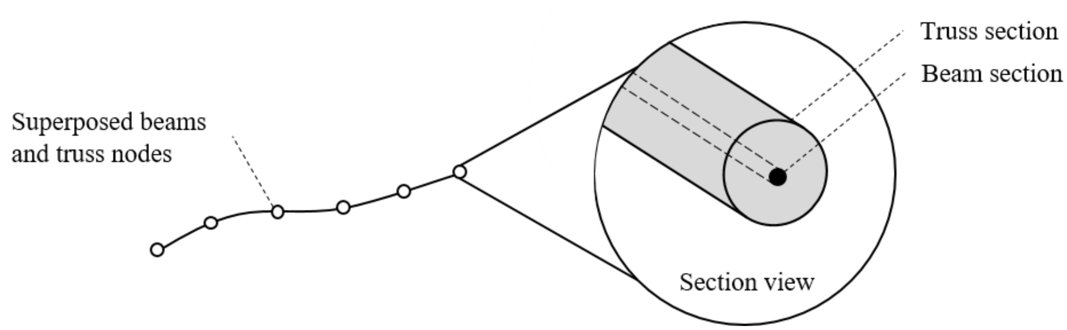

Figure 1: Representation of a section of the embedded filaments, the central beam is used for its contribution to the bending stiffness while the external truss elements are use for their contribution to tensile stiffness and contact resolution

The idea behind the use of embedded beam elements to model filament bundles is that when a dry tow is bent, the bending stiffness should be close to the one of a single filament. Since they are free to move, the filaments can easily spread. Their contribution to the bending stiffness is therefore highly decreased. However, every single filament contributes to the tensile stiffness and therefore this contribution can be modelled by the equivalent section of a truss element. This embedded structure (see Figure 1) is achieved by superposing two meshes, one made of truss elements, and one made of beam elements. All the degrees of freedom (d.o.f.) of the superposed nodes are then linked using multi-points constraints.

2.1 Braid geometry and material

The braid considered in this work is a 12 tows braid with 100 filaments per tow. Each tow initially forms a 55° angle with the braid axis. The braid's internal radius is 3mm. The fibers used were chosen to match the mechanical properties of commercial glass fibers (ECD900MM 620-1, AGY). The mechanical properties used are reported in table 1.

Table 1 Mechanical properties used for the glass fibers

Mechanical property	**Value**
Young Modulus	72 GPa
Poisson Ratio	0.23
Density	2.58 g.cm^3
Filament radius (beam elements)	2.5 µm

2.2 Boundary conditions

The model presented here corresponds to a representative section of a longer braid. In order to accurately represent its behaviour, it is necessary to use periodic boundary conditions (PBC). These conditions ensure that both ends of the braid are virtually connected, therefore effectively representing an infinitely long braid. These conditions can be implemented by using linear equations. One needs to introduce a so called "dummy node" u^{Dum}. Let u_i^P (respectively $u_i^{P\prime}$) be the nodal variables corresponding to the degree of freedom $(d.o.f.)$ i and the node P (respectively its corresponding node on the other side P'), then the periodicity of the displacement field can be expressed as:

$$u_i^P - u_i^{P\prime} + u_i^{Dum} = 0, \forall\, i \in \{d.o.f.\} \quad (1)$$

It is therefore possible to impose a periodic strain by imposing a value on any degree of freedom of u^{Dum}.

3 MODEL VALIDATION

Three main parameters are studied in this work: The braid thickness, the angle between the tows and the braid axis and finally the final length of the braid. To validate the convergence, the evolution of these three parameters was studied using four mesh refinements: 31, 61, 91 and finally 121 elements per filament. The relative change between two steps is defined as:

$$R_c = 100 \frac{|a^{i-1} - a^i|}{a^i} \quad (2)$$

where a^i is the value taken by the observed parameter at the ith convergence iteration. Convergence is considered achieved when R_c gets lower than a 95% trust interval, which can be written as

$$R_c < 100\frac{2\sigma}{a^f\sqrt{n}} \tag{3}$$

where σ is the averaged standard deviation, n is the number of measurements and a^f is the value obtained for 121 elements per yarn. Here, $n = 6$ for every parameter. An exception had to be made for the length convergence since no trust interval can be defined as the PBC intrinsically impose that the distance between the extremities of the filaments will be exactly the same thorough the whole model. In this case, an arbitrary value was chosen as 0.5%.

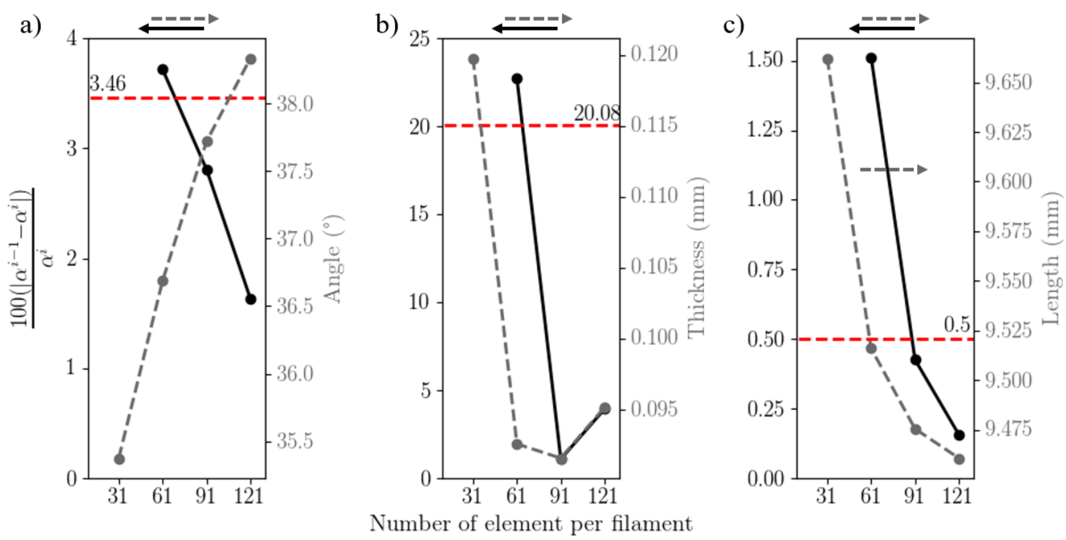

Figure 2: Convergence study of the reference case. The black in each graph corresponds to the relative change between two iterations, the gray, dashed, line corresponds to the measured values and the red-dashed line corresponds to the convergence criterion value as defined in equation 2. a): Angle convergence, b) Thickness convergence, c) length convergence.

The Figure 2. displays the evolution of R_c for every parameter as a function of the number of filaments per tow. The Figure 2a) corresponds to the angle relative changes, the Figure 2b) to the thickness, and finally, the Figure 2c) displays these results for the length. The red-dashed line corresponds to the convergence criterion as defined in eq. 2, the black line corresponds to the relative change between two iterations and the gray-dashed line corresponds to the measured values. For every parameter, the convergence behaviour is well defined as R_c exhibit a strong negative slope. Based on the results shown in Figure 2. It was decided to use the 91 elements per filament configuration for the rest of this study.

4 RESULTS

4.1 Visual comparison

The first validation step is to qualitatively compare the results obtained. The Figure 3 shows the different results obtained when the different braids are tighten on the analytical solid using horizontal tension. The truss elements (Figure 3c) result displays a strong lack of bending stiffness, which resulted in the noisy disposition of the filaments. Many small curvatures can be observed making it visually very far from the reference. The beam elements (Figure 3b) results are much closer in terms of overall organization,

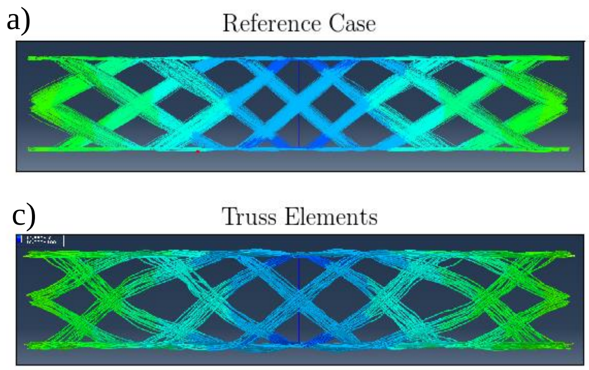

Figure 3: Comparison between the reference case and the different types of elements a): Reference case, 100 filaments per tow, beam elements matching the real filaments. b) Beam elements, 7 filaments per tow. c) Truss elements, 7 filaments per tow. d) Embedded elements, 7 filaments per tow

yet one can observe that in the corners of the analytical solid these elements are too stiff. This is not surprising considering that their radius is much larger than the real filament radius. Finally, the embedded elements (Figure 3d) solution seems to be the closest to the reference case. In this case, the individual filaments seems to exhibit enough stiffness to remain straight on the faces of the analytical solid yet still have the ability to bend on its corners.

4.2 Angle comparison

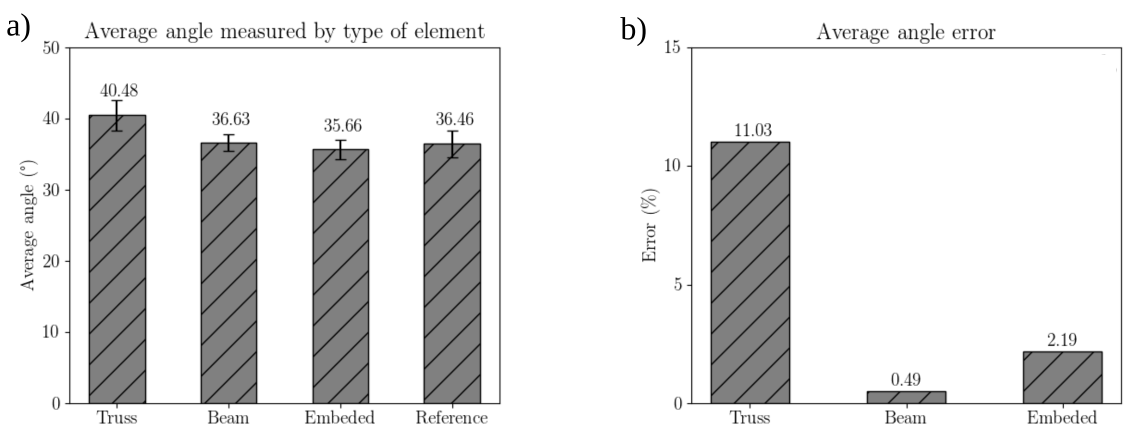

Figure 4: a) Average angle (°) measured in the different models, the error represents a 95% trust interval. b) Relative avergage error (%) between the 7 filaments model and the 100 filaments reference

The Figure 4a) displays the average angle measured within all the models. Both the beam and embedded models gives reasonable estimations of the angle as they are both within the error bars of the reference. The truss model estimate is however slightly too large. This can probably be attributed to an increase in friction between filaments as the the truss solution is very chaotic.

Figure 4b) shows the same results, but expressed as relative error (%) between the 7 filament cases and the reference. Similarly to what can be seen in Figure 4a), the beam and embedded estimates are better than those of the truss model, with an error 5 times lower for the embedded model and 22 times lower for the beams. Overall, the data displayed in Figure 4 tends to show that truss element would not be appropriated to model braids. It is however impossible to know which model is better between beams and embedded with only the angle data as their estimation fall under the boundaries of the 95% trust interval.

4.3 Thickness comparison

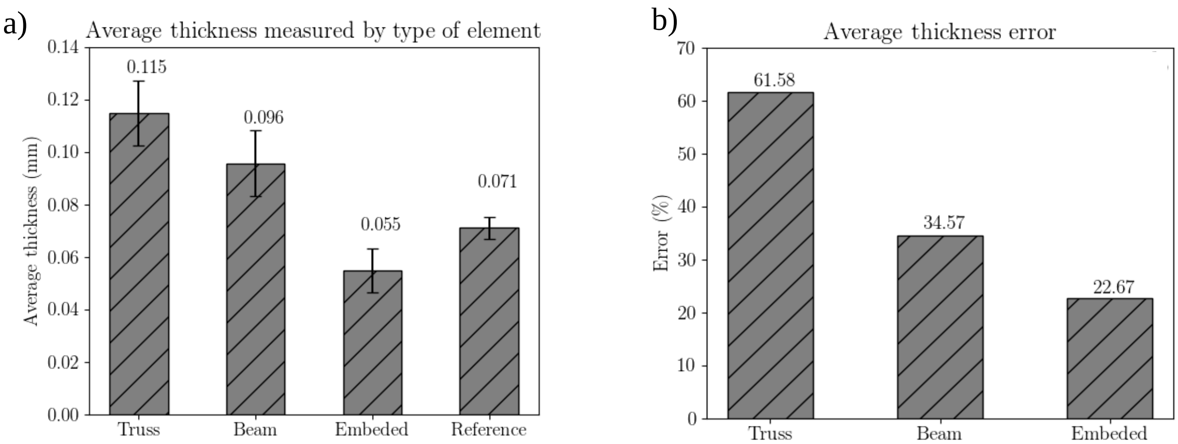

Figure 5a) Average thickness (mm) measured in the different models, the error represents a 95% trust interval. b) Relative avergage error (%) between the 7 filaments model and the 100 filaments reference

Figure 5a) shows the average thickness (mm) measured in the different models. This thickness was measured in six different positions corresponding to where the individual crossover. The error bars corresponds to the 95% trust interval, Figure 5b) shows the average error when comparing the 7 filament solutions to the 100 filament reference. The worst estimation comes from the truss elements. This high error is most likely caused by the lack of stiffness. The noisy distribution preventing filaments from having a smoother organization and therefore creates a thicker bundle. The beam results also overestimate the thickness. This is most likely attributable to the overestimation of the bending stiffness. The embedded elements estimation is better in terms of average error but it is underestimating the thickness. This can be explained by the better bending ability of the embedded elements when compared to beams while keeping a relative bending stiffness which prevents noise. It is likely than by adding more and more filaments, the beam results would converge by decreasing and the embedded element would converge by increasing.

4.4 Lenght comparison

The last parameter that was studied in this work is the braid's length at the end of the virtual experiment. This parameter can be associated to the effective stiffness of the braid as a constant force is applied thorough the model run. The Figure 6a) shows the length (mm) measured between both ends of the filaments compared by type of elements, and Figure 6b) shows the relative error (%) when comparing these lengths to the reference case. Here, the problem associated with the beam too high stiffness is very visible. Indeed both the truss and embedded elements results gives estimation within a 5% error while the beams gives a very under-estimated length with a nearly 20% error. It is also interesting to note that the embedded elements are able to almost exactly estimate the final length with an error of 0.33% when compared to the reference solution.

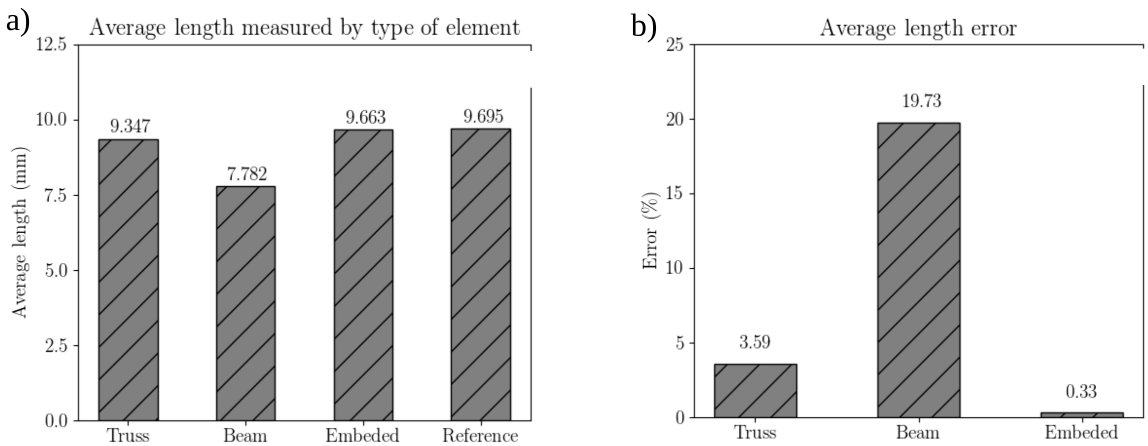

Figure 6a) Length measured from one end to another of each filament. There is no error bars in this plot since the length is imposed as the same by the PBC. 6b) Average length error when comparing the 7 filament estimations with the 100 filament reference

4.5 Calculation time and cumulated error

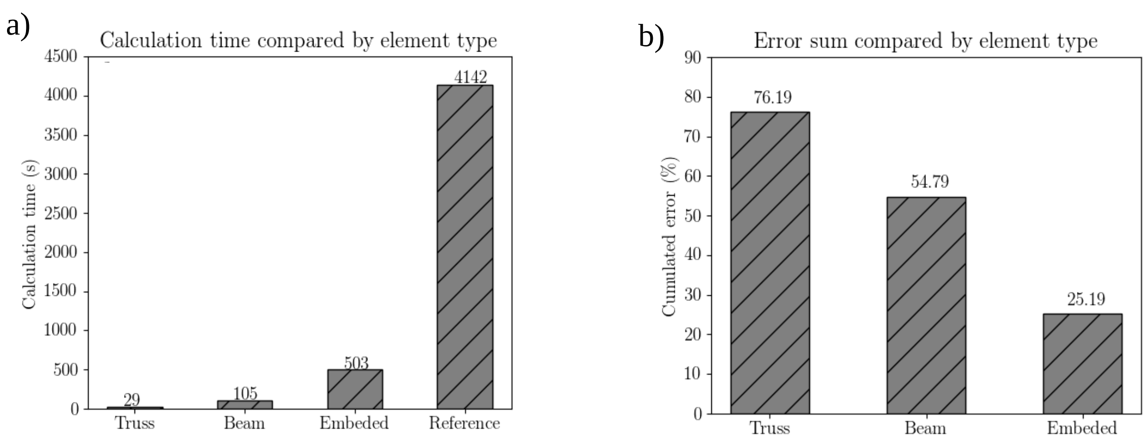

Figure 7 a) Calculation time of each case presented in this paper b) cumulative error of each 7 filament model when compared to the reference solution

Figure 7a) shows the calculation time of each model, and Figure 7b) shows the cumulated error for each model (sum of the values displayed in Figure 4b, 5b and 6b). The truss model is the one that runs the faster, it is able to give an estimation in 29 seconds. It is, however, the model that is the farthest from the reference, with a cumulated error of 79.19%. The Beam model comes second, with a running time of 105 seconds. It is still relatively far from the reference with a cumulated error of 54.79%. The slowest model is also the closest to the reference, it is able to reach a cumulated error of 25.19% while still being more than 8 times faster than the reference. This probably means that the embedded model would converge faster when adding more filaments in the cross-section.

CONCLUSION

We propose in this work, a way of accurately modelling filament bundles. The methodology consists in using embedded elements to model the different contributions of the filaments individually and as a group. Using a small beam element in the center allows an accurate modelling of the filament bending stiffness while a larger rod element models the longitudinal stiffness of the bundle. A 7 filament bundle was compared to a 100 filament reference case (using an equivalent total cross-section area) and the embedded approach proved to be better than a full beam or truss approach. A limit of this approach could be that the gain in precision is achieved by effectively doubling the number of nodes and therefore could induce a lesser calculation time over precision ratio than traditional approaches. Future works will be dedicated to the study of the convergence rate of each approach (as a function of the number of filaments used to model a full bundle) as well as the calculation time over precision ratio with different configurations. An experimental confrontation is also planned as future work.

REFERENCES

Castañeda, P. P. and J. R. Willis (1999). "Variational second-order estimates for nonlinear composites." Proceedings of the Royal Society of London. Series A: Mathematical, Physical and Engineering Sciences **455**(1985): 1799-1811.

Döbrich, O., T. Gereke, et al. (2016). "Modeling the mechanical properties of textile-reinforced composites with a near micro-scale approach." Composite Structures **135**: 1-7.

Feyel, F. (2003). "A multilevel finite element method (FE2) to describe the response of highly non-linear structures using generalized continua." Computer Methods in Applied Mechanics and Engineering **192**(28-30): 3233-3244.

Fish, J. (1992). "The s-version of the finite element method." Computers & Structures **43**(3): 539-547.

Fish, J., S. Markolefas, et al. (1994). "On adaptive multilevel superposition of finite element meshes for linear elastostatics." Applied Numerical Mathematics **14**(1-3): 135-164.

Ghossein, E. and M. Lévesque (2014). "A comprehensive validation of analytical homogenization models: The case of ellipsoidal particles reinforced composites." Mechanics of Materials **75**: 135-150.

Mori, T. and K. Tanaka (1973). "Average stress in matrix and average elastic energy of materials with misfitting inclusions." Acta Metallurgica **21**(5): 571-574.

Moulinec, H. and P. Suquet (1994). "A Fast Numerical-Method for Computing the Linear and Nonlinear Mechanical-Properties of Composites." Comptes Rendus De L Academie Des Sciences Serie Ii **318**(11): 1417-1423.

Moulinec, H. and P. Suquet (1998). "A numerical method for computing the overall response of nonlinear composites with complex microstructure." Computer Methods in Applied Mechanics and Engineering **157**(1-2): 69-94.

Nilakantan, G. (2013). "Filament-level modeling of Kevlar KM2 yarns for ballistic impact studies." Composite Structures **104**: 1-13.

Nilakantan, G., M. Keefe, et al. (2010). "Multiscale modeling of the impact of textile fabrics based on hybrid element analysis." International Journal of Impact Engineering **37**(10): 1056-1071.

Tabatabaei, S. A., S. V. Lomov, et al. (2014). "Assessment of embedded element technique in meso-FE modelling of fibre reinforced composites." Composite Structures **107**: 436-446.

Wang, Y., Y. Miao, et al. (2010). "Digital element approach for simulating impact and penetration of textiles." International Journal of Impact Engineering **37**(5): 552-560.

Zhou, G., X. Sun, et al. (2004). "Multi-chain digital element analysis in textile mechanics." Composites Science and Technology **64**(2): 239-244.