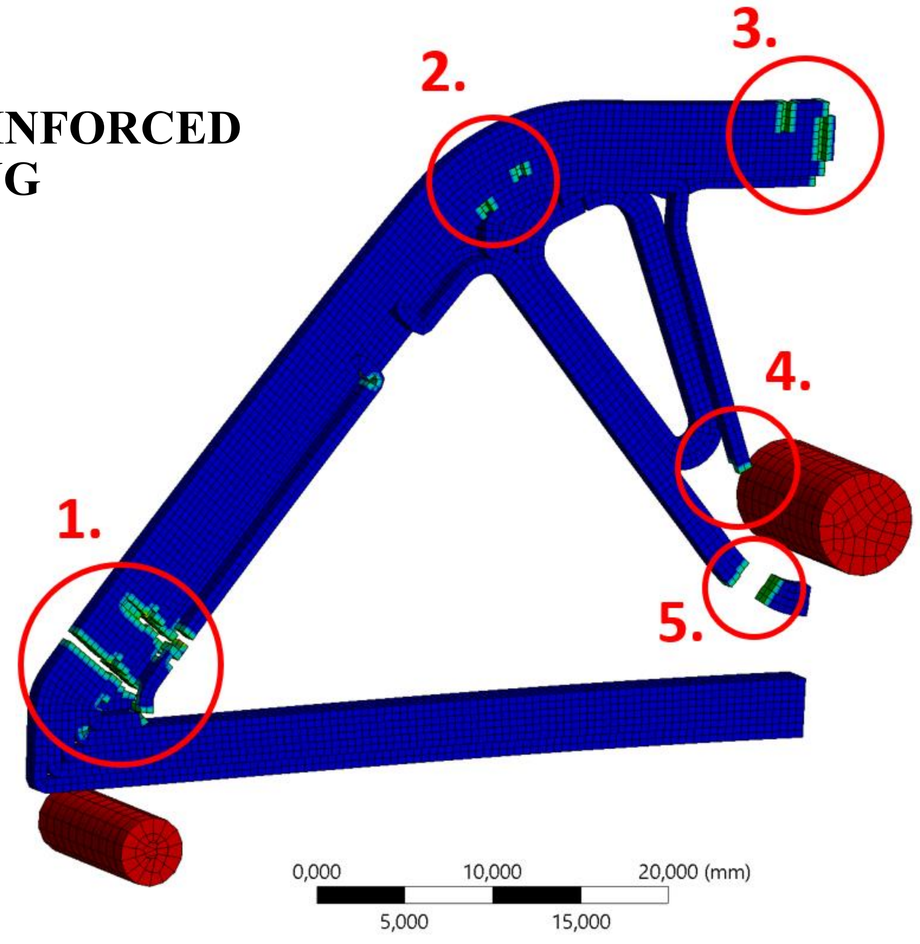
FINITE ELEMENT MODELLING OF CONTINUOUS FIBER–REINFORCED COMPOSITES PRODUCED BY AUTOMATED MANUFACTURING



APPLICANT: BENCE SZEDERKÉNYI

CONSULTANT: DR. TIBOR CZIGÁNY, DR. NORBERT KRISZTIÁN KOVÁCS

INSTITUTION: BUDAPEST UNIVERSITY OF TECHNOLOGY AND ECONOMICS

FACULTY: POLYMER ENGINEERING

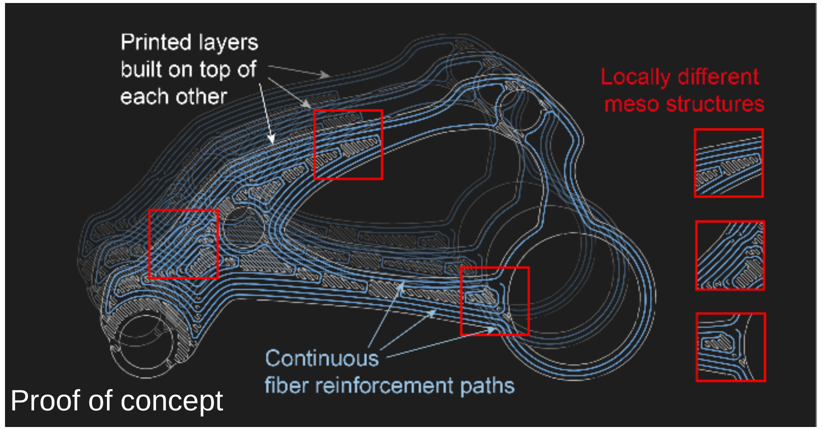
E-MAIL: szederkenyib@pt.bme.hu

PHONE: +36304144709

INTRODUCTION AND PROJECT OBJECTIVE

Introduction

With the advent of automated, continuous filament placement technologies, the possibilities for reinforcement placement in composite manufacturing have been further expanded, as the reinforcement path can be continuously varied within the layer. This allows far more efficient structures to be created, reducing the structure's weight compared to conventional composites. At the same time, this manufacturing freedom also poses a significant challenge for the structural design of composite components, as simulation methods based on meso-level homogenization are mostly not usable due to the lack of periodicity of the reinforcement structure.

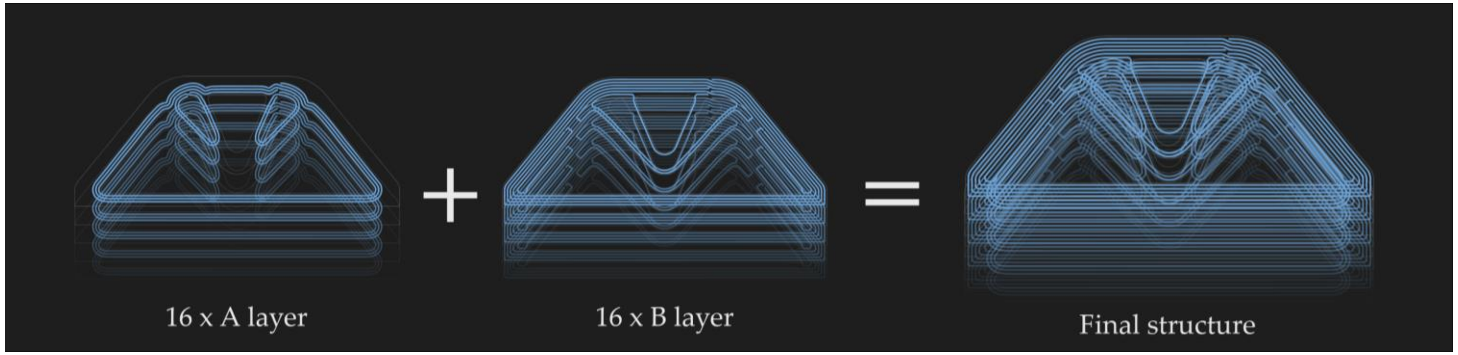


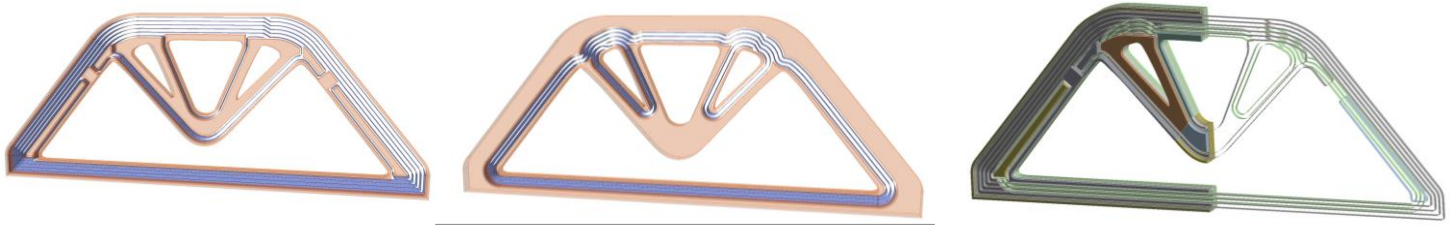
Project objective

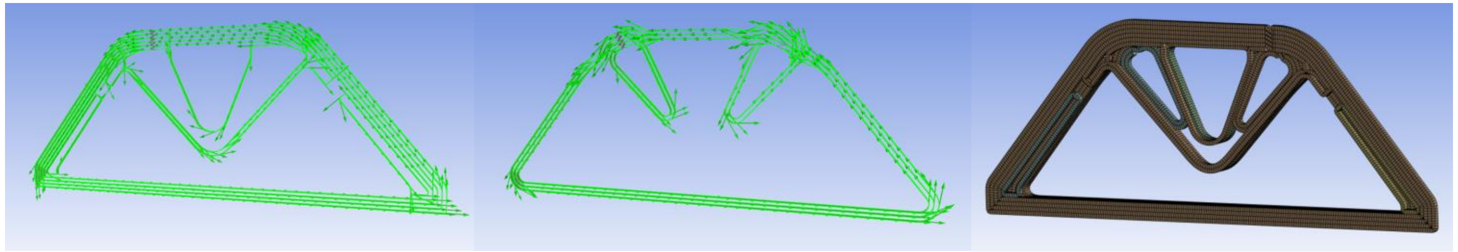
In this project, I aimed to develop a procedure for structural analysis based on finite element methods (FEM) for 3D printed, continuous fiber-reinforced composites, including the modelling of progressive failure mechanisms, like the gradual damage of fiber bundles and interlayer delamination processes.

MODEL STRUCTURE, DISCRETISATION

- Ansys Workbench 2023R2 environment.
- two separate characteristic reinforcement structures
- Reinforcement structure was prepared in Ansys Composite PrePost, the matrix in Static Structural, then merged in a shared module, where the final loading and contact boundary conditions were determined
- Based on the edges of the central surfaces representing the fiber reinforcement fiber directions were determined, the central surface was symmetrically thickened according to actual fiber thickness (solid composite modelling)

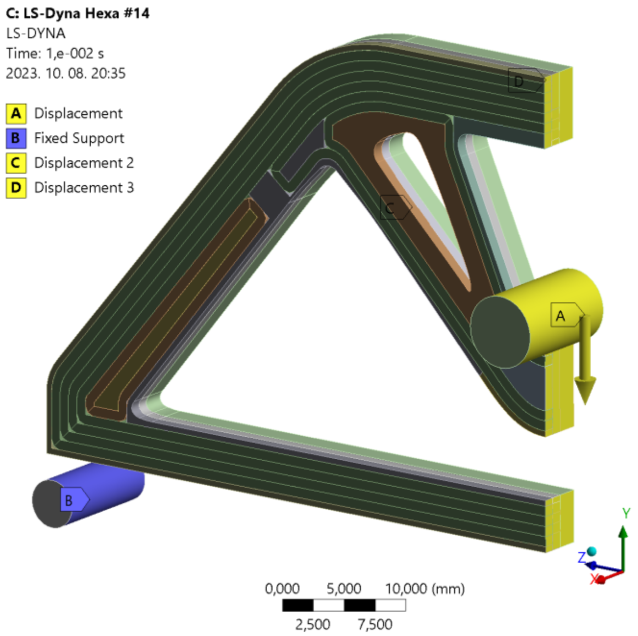


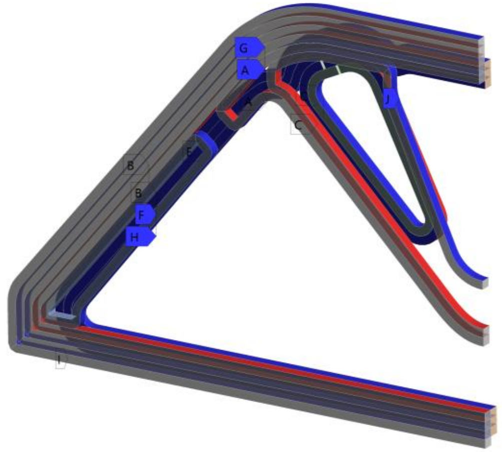


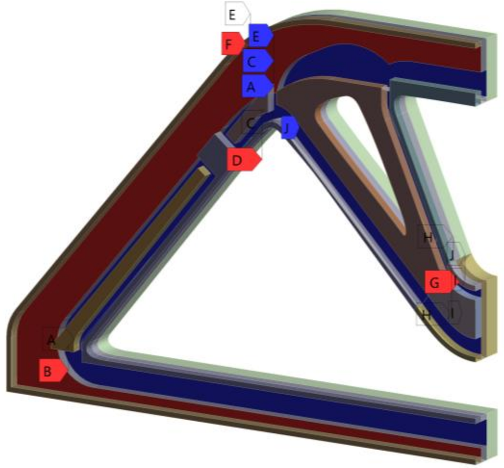


TYPE OF CALCULATION, APPLIED LOAD AND BOUNDARY CONDITIONS

- Explicit structural FEA, **Ansys LS-Dyna**
- Between specimen's components normal- and shear stress limited bonded constraints
- External parts connected to the specimen with body interaction constraint with a coeff. of friction: static – 0.1; dynamic – 0.05
- 7 mm vertical displacement through a pin
- Symmetrical support placed with an 80 mm span distance
- Modelled as a quarter model, frictionless planar constraint in the symmetry planes
- Modelled with higher strain rate, since the applied ***MAT54** material card, not a strain dependent one
- 0.5 mm element size, timestep of 4.5×10^{-5} ms, adaptive mass scaling with a 1.7% increase in total mass
- Mesh constructed from linear hex8 and wed6 elements with an average quality of 0.9483±0.082 according to the built-in Ansys metrics
- Runtime at full processor use was ~2 hours per iteration on an Intel I7-7700K, 4.2 GHz

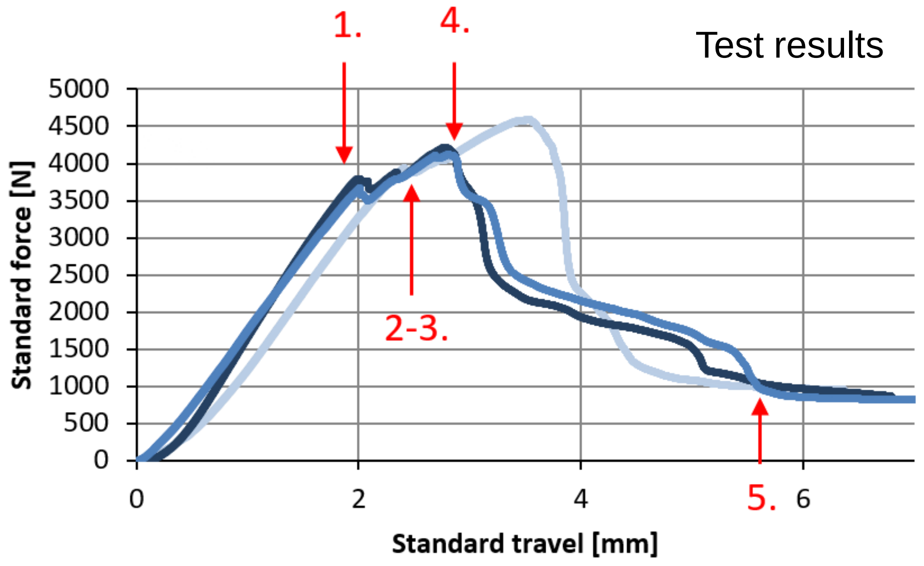


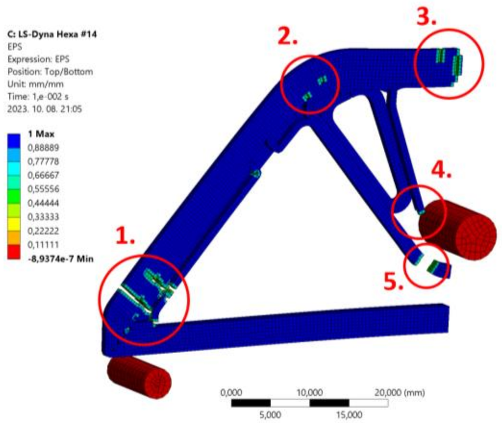


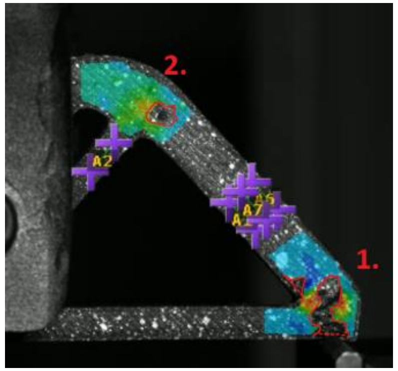


EMPLOYED MATERIAL CARD AND TUNING

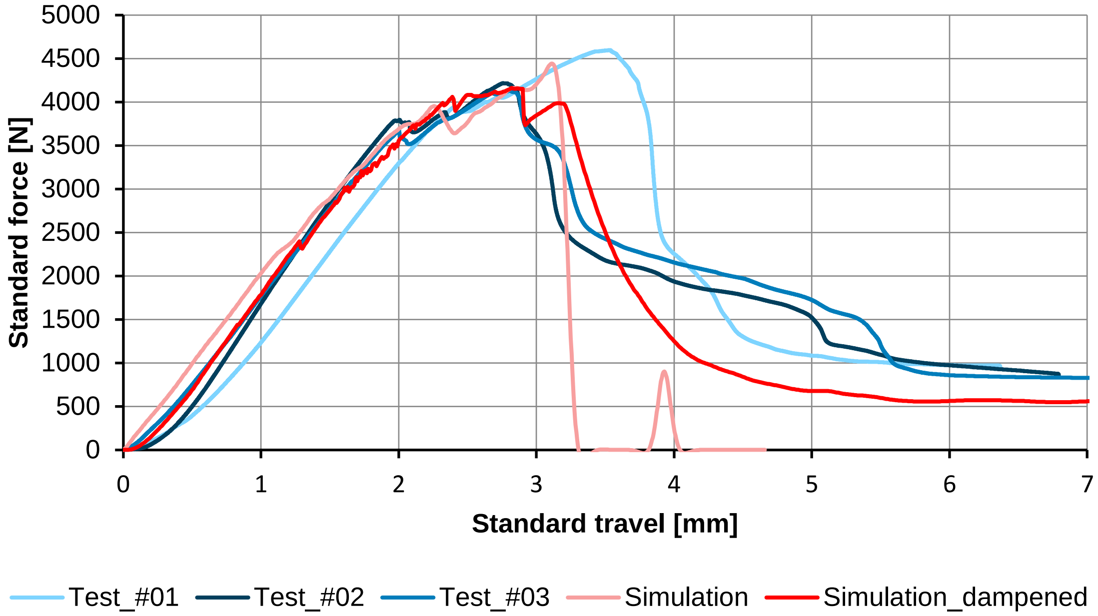
- Reinforcement and matrix: ***MAT54**
- Starting model based on literature and the remaining parameters based on RVE modelling
- Material cards tuned by the presented simulations in 51 iterations
- ***Idea:*** *the different sections of the complex measurement curves dominated by the effects of the various engineering constants*
  - *moduli can be tuned based on the initial linear section.*
  - *Failure limit values can be obtained based on the local force maxima of each force peak*
  - *co-occurring, redundant effects are difficult to separate from each other and cannot be tuned independently. This can be addressed by performing a few more parallel simulation validation on similarly complex but independent problems.*
- Global dampening (with a value of 250000) added after the 45. iteration for addressing the viscoelastic dampening in the system

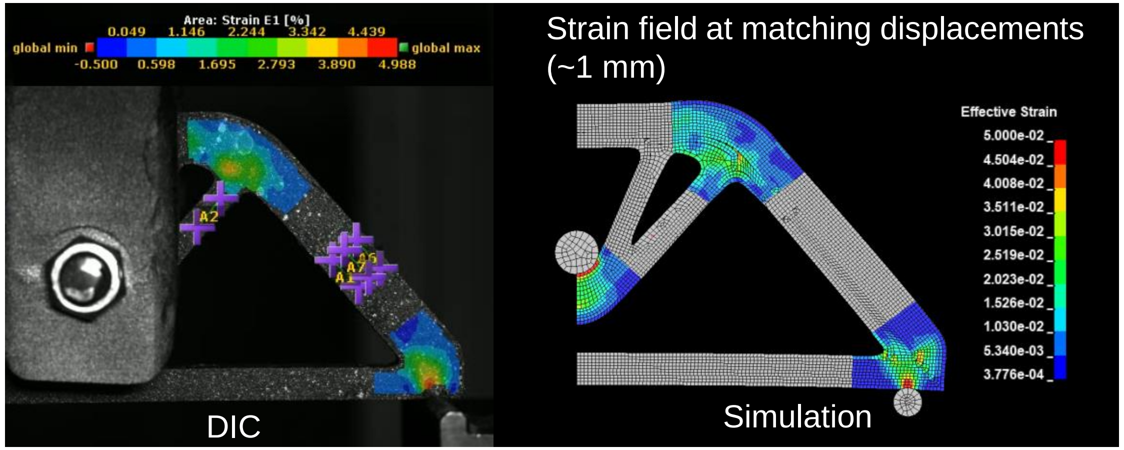




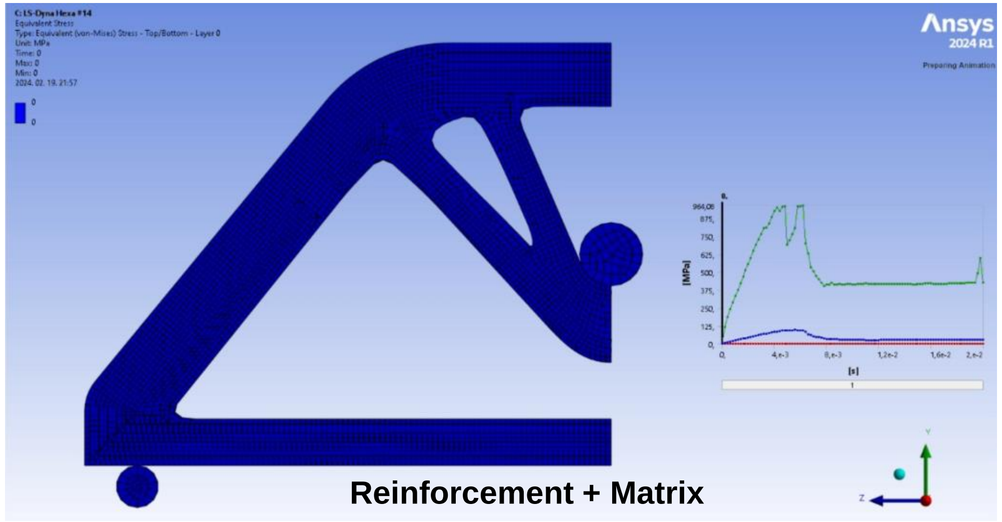


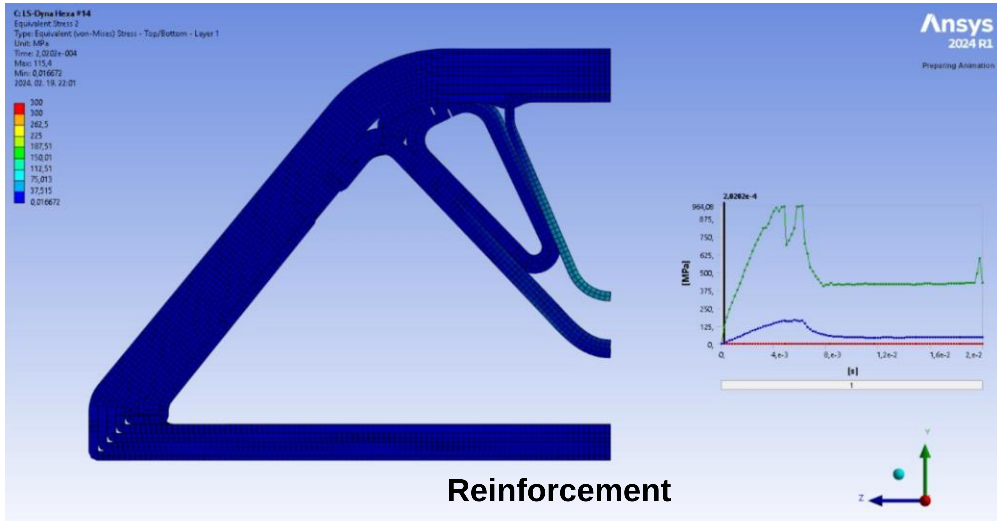
RESULTS AND EVALUATION





Structural FEA response





CONCLUSIONS

1. Simulations and validation tests were performed on a topology-optimized test specimen subjected to three-point loading with complex inner load states.
2. Verification tests were performed with the addition of a DIC system, the results of which were later used to explore the local failure characteristics and finally compared with our simulation results.
3. Constitutive parameters of the initial material model were partly taken from literature with supplementary tests and RVE modelling.
4. Constants with only mathematical meaning were determined with a heuristic, iterative method in 51 iterations.
5. The results demonstrate a promising correlation with the measured values, especially supplemented with a global dampening factor.

Overall, the presented method provides a suitable modelling basis for continuous fiber-reinforced composites produced by automated processes, thus significantly improving the engineering applicability of these structural materials and manufacturing methods.