Numerical Modeling of Ultrasonic Inspection in Fiber Reinforced Materials with Explicit Microstructure

Andreea-Manuela ZELENYAK [1], Reinhold OSTER [2], Michael MOSCH [2],
Peter JAHNKE [3], Markus G. R. SAUSE [1]
[1] University of Augsburg, Institute of Physics, Augsburg, Germany
[2] Airbus Helicopters Deutschland GmbH, Donauwörth, Germany
[3] BMW AG, München, Germany

Contact e-mail: andreea-manuela.zelenyak@physik.uni-augsburg.de

Abstract. For materials, such as fiber reinforced composites, the explicit microstructure renders attempts to perform quantitative ultrasonic inspection more difficult than for isotropic materials. The textile architecture of modern composites includes stackings of fiber oriented to different angles, combines woven fabrics and plies with straight filament orientations and includes warp and weft rovings as binder materials. Ultrasonic inspection is meant to provide a reliable information on the presence of defects, such as porosity, cracks or delamination, but suffers in interpretation by the interaction of the incident ultrasonic beam with the material microstructure in the same way as with defects. To overcome limitations in ultrasonic signal interpretation, this presentation presents some latest results to perform numerical modeling of ultrasonic inspection in such scenarios. To build a realistic model, the geometry of the individual plies, rovings and defects are extracted from a computed tomography scan, are digitally refined and are incorporated into a finite element model. Applying a 3D multi-scale and multi-physics computation this approach allows to obtain modeled signals, which can be quantitatively compared to experimental signals. This is possible by including all aspects of signal conversion in the modeled probehead as well as the influence of the attached circuitry by simultaneous electrical simulations.

1. Introduction

One commonly used Non Destructive Testing (NDT) method used to investigate composite material parts is Ultrasonic Testing (UT). Even though being such a well-known technique, reference standards for equipment calibration and material characteristics are still not fully established. Inspection of simple geometries including defects is already established but the relation of equipment sensitivity to the defect shape, orientation, material microstructure, realistic crack patterns, etc. is still under investigation. Modern composite materials can be made from different fiber materials but also can have different fiber architectures like rovings, combining unidirectional layers and fabric in accordance with design requirements. Interaction between the UT signal and such a complex material structure is still under experimental investigation. In order to optimize the NDT process or to increase

More info about this article: http://ndt.net/?id=19482

the cost efficiency, accompanying modeling of the UT has lately been established to aid the user before performing experiments.

Ultrasonic inspection simulation is possible using different commercial software packages which provide tools to predict the beam of a transducer in the inspected specimen or to predict the interaction of the beam with the defects. Computation of the A- B- and C-scans is already possible using semi-analytical kernels, ultrasonic tools being created in order to conceive, optimize and predict the performances of various inspection techniques [1]. However, the accuracy of the models generated with such software is limited by using the semi-analytical mathematical formulation approach [2]. Usually in such approaches the sample geometry does not include a detailed microstructure or the exact geometry of the embedded defects.

To consider these details, the finite element method (FEM) can provide a better approach for modeling ultrasonic signal propagation in composite materials fully accounting for the complexity of the material during wave propagation. Due to the continuous development of commercial Multiphysics platforms such as Comsol, Ansys or Abaqus and the advent of high-power computer technology, FEM can readily be used to solve complex multi-physics problems. In the current context, this includes the full chain of ultrasound generation and detection, wave propagation and the interaction of the wave with internal defects. Finite element simulation of ultrasonic wave propagation and its interaction with defects have been studied e.g. by Ludwig and Lord [3].

A realistic model in case of ultrasonic testing of a composite material would not solely include the sample geometry but also the transducer. Since the characteristic of the transducer will influence the detected signals, this needs to be included in the model. To this end, fully coupled multi-physics approaches have been proposed to consider the details of the sensor by explicit modeling of the sensor geometry in combination with attached circuitry [4, 5]. To capture the detailed shape and position of internal defects a procedure to extract geometries from computed tomography (CT) scans has been proposed as well [6].

The aim of this paper is to present results of a 3D multi-scale and multi-physics model for ultrasonic testing of a composite material with explicit microstructure. To this end, the composite material structure is extracted from CT images and is implemented in a numerical model as geometry for the samples subject to UT. We use Comsol Multiphysics as modeling environment, since it allows direct multi-physics coupling of the description of wave propagation (structural mechanics), signal detection (piezoelectric effect) and the influence of the attached electronics (P-SPICE). For this investigation the operation mode for ultrasound inspection equipment was chosen as pulse-echo mode. In this configuration the transducer emits the signal and it also receives it. The presented approach thus allows to perform a quantitative comparison between modeling and experimental results.

2. Finite element modeling of ultrasonic testing

A quantitative approach to model ultrasonic testing simulation using FEM contains two steps. The first step is to validate each part of the model by using experimental results. In the present case, this will include a realistic description of the transducer, a correct choice of material properties of the composite sample and an accurate description of the interaction with embedded defects. In the second step, such a validated model environment may then be used to perform parameter studies. This is to analyze the impact of the material microstructure and the interaction of embedded defects like pores, delamination, cracks, particle inclusions, or fiber waviness on the ultrasonic signal.

2.1 Validation of the modeling approach

The approach we have chosen to validate our simulation model is called *multilevel model validation* and a scheme presenting the steps used to validate the FE model can be seen in Figure 1. There are four levels used for validating the UT simulation work: *geometry, material properties, physics and solution convergence*. At each level the validation process is required for the transducer but also for the sample under investigation.

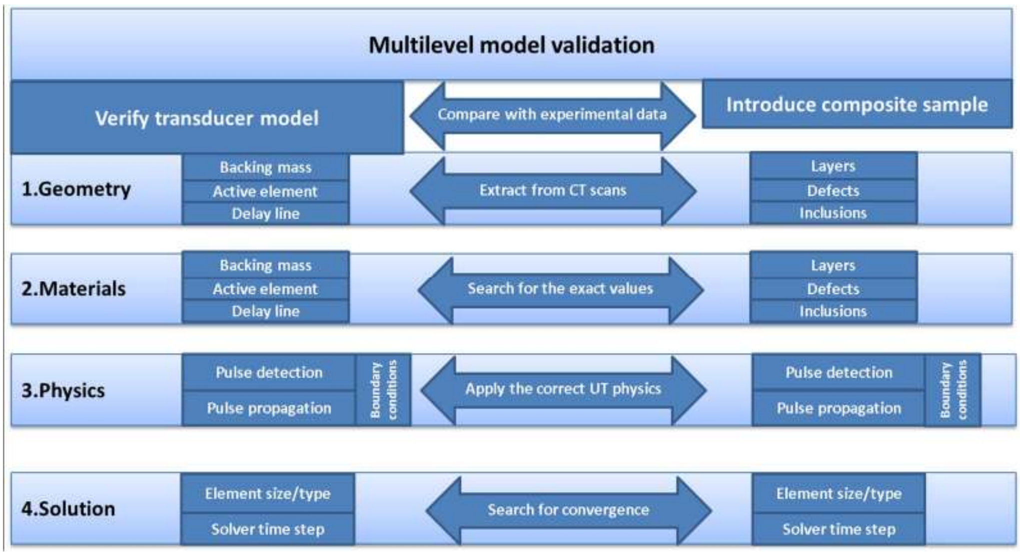


Fig. 1. Multilevel model validation scheme

2.2 Geometry implementation

The first step is to include a realistic geometry of the transducer with all internal components. The ultrasound transducer model was chosen to be V201 RM model from Olympus which represents a typical single element contact transducer with a delay line. All dimensions of the geometry were extracted from a computed tomography scan of the transducer, individual parts being identified based on their different material densities.

In Figure 2 a scheme of the geometrical implementation of the transducer is presented. All identified parts are presented with different colors shown as cross section of the transducers geometry. The absolute dimensions of the parts were systematically varied, taking into the account the uncertainty of the CT measurement and to assess the impact upon the detected signals and to fit with the experimental measurements. As an example, only a small variation of 400 µm of the piezoelectric element radius was found to induce significant increase in amplitude of the detected signal.

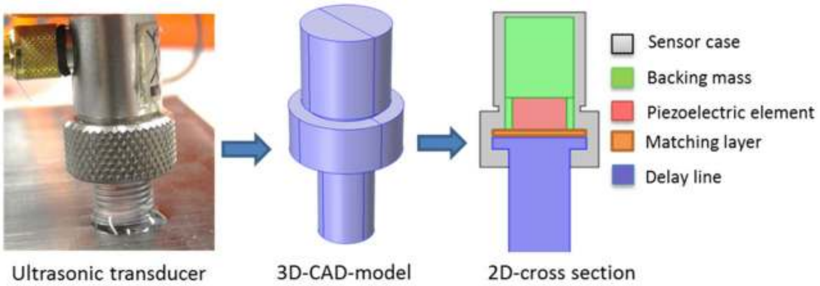


Fig. 2. Typical ultrasound transducer implementation in a model geometry.

To increase the modeling efficiency an axial symmetric implementation is used for the transducer model. Some parts of the sensor were not modeled (e.g. sensor casing), all simplifications being concluded from running study cases and comparing the changes to signal propagation with the validated one.

The second step in geometry implementation is the representation of the test laminate including the necessary level of detail. Again, the CT scans are used in reconstructing the model geometry of the investigated sample. This especially includes the extraction of the embedded defects as 3D objects from the measurement. The work-flow to achieve this implementation is shown in Figure 4.

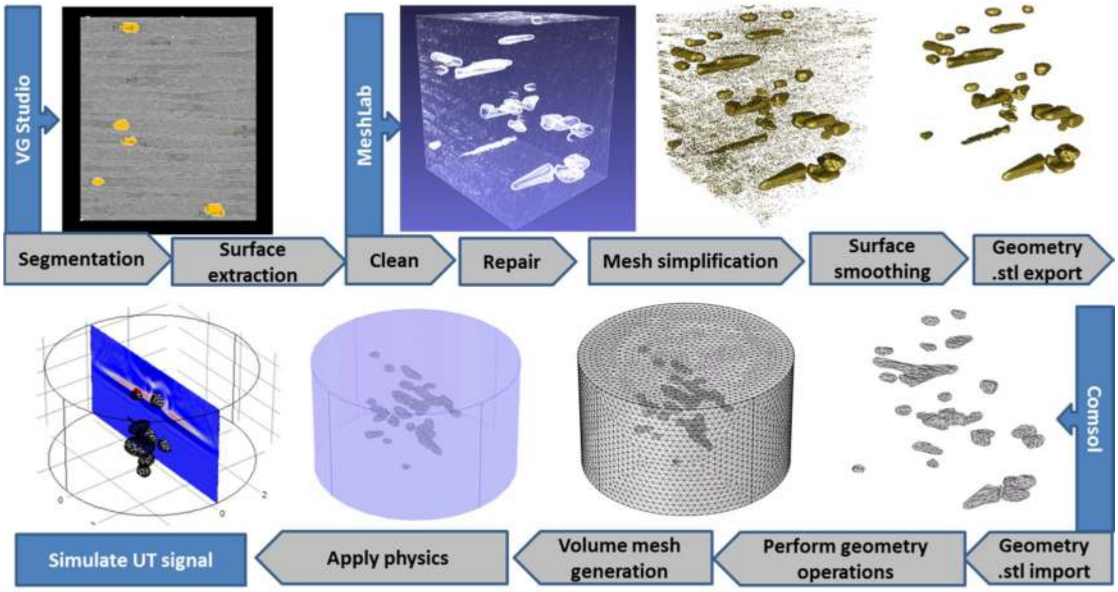


Fig. 3. Workflow to implement geometry from CT scans in Comsol Multiphysics [7].

The CT scan of the sample probe is a typical multi-axial composite material with embedded pores [7]. The presence of porosity can be seen in the CT cross section presented in Figure 3 (marked in orange).

In this investigation the geometry extraction and the FEM implementation of porosity as characteristic type of defect is presented. Two software packages are required before being able to import such a geometry in the Comsol Multiphysics environment. For the CT scans VGStudio MAX 2.1 is used as a software solution for voxel data analysis, visualization and 3D volume reconstruction. Segmentation of the pores is done based on an threshold routine classifying the gray scale values into regions with similar level of color, texture, contrast or brightness based on histogram features [8].

After the segmentation of the pores, their surfaces are extracted using the Surface Extraction tool included in VGStudio MAX 2.1. This results in a triangular mesh surface, which is exported in a stl file format. The mesh extraction parameters can be varied in terms of the mesh quality. For the present study a “normal” quality of the mesh network was found to work well with the extraction but also with the further CAD import procedure in Comsol Multiphysics.

However, even when using high resolution CT scans there are imperfections of the scan, such as inhomogeneous intensity regions or imaging artefacts. As a result, the extracted mesh suffers from inclusions, holes or a lack of node integrity. Thus a direct import of these mesh surfaces in Comsol Multiphysics is not recommended.

Instead, the open source software MeshLab is used in-between for preprocessing the exported mesh. Different filters are used to clean and repair the mesh structure. The most important ones were those removing the non-manifoldness of the mesh network and those closing the open holes of the surface. To increase the efficiency of the computation, simplifications of the mesh network are also applied using the quadratic edge collapse method with initial topology preservation. The resulting pore geometry are then imported in Comsol Multiphysics and are used as defect geometry. Details of this implementation step can be seen in Figure 3. To represent the sample laminate, the individual stacks and the full volume of the material are created in the integrated CAD environment of Comsol Multiphysics and are connected to the defect models by operations like intersections or unions.

2.3 Material properties

Material properties used for the simulation were extracted from manufactures datasheet and literature. The values used for the sensor implementation were systematically varied to investigate the impact upon signal generation and propagation. The most important properties for the sensor case were those describing the anisotropic piezoelectric element and the backing mass, which finally were imported from the Comsol Multiphysics material library. The properties required for describing a typical composite sample are the density and the elasticity matrix of a homogenized unidirectional layer. To calculate the full elasticity matrix for an anisotropic material, which exhibits transversal isotropy behavior, five constants are necessary: E_{11}, E_{22}, ϑ_{12}, ϑ_{23}, G_{23} representing the modules of the principal directions, Poisson’s ratios and the in plane shear modulus.

2.4 Applied physics and convergence of solution

The third and the fourth step in the model validation work consist of the selection of a physical description of the problem and the check for conversion of the solution.

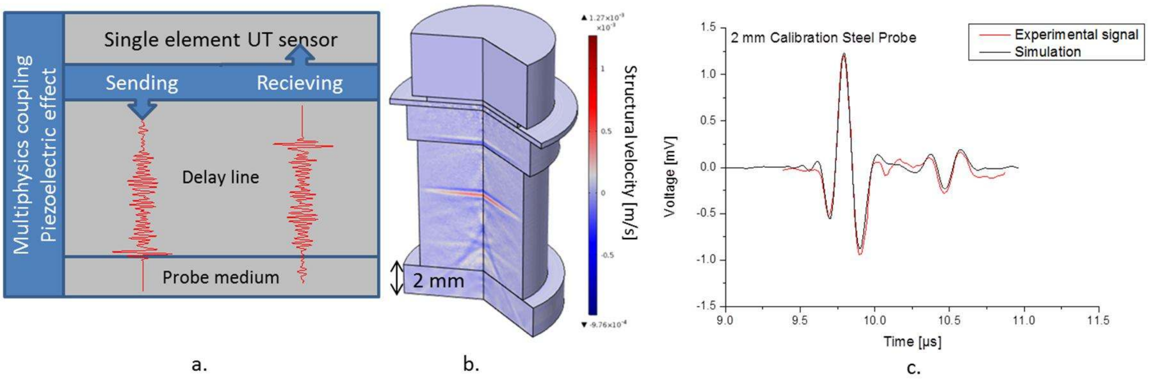


Fig. 4. a. Schematic representation of the modelling approach **b**. Image of wave propagation at 7 µs **c.** Comparison of modeling result with the experimental signal obtained on steel calibration block.

Within the AC/DC module of Comsol Multiphysics an electrical excitation pulse with 5 MHz frequency is generated and is applied as voltage source to the active piezoelectric element through a P-SPICE circuit simulation. The conversion of this electrical signal into an ultrasonic wave is then accounted for by solving the coupled piezoelectric equations in the transducer element. The generated ultrasonic wave propagates along the delay line and into the sample. The wave propagation is defined within the equations of the Structural Mechanics module. This fully accounts for the properties of the propagation medium and the interaction of the wave with the boundaries or defects. The reflected ultrasonic wave is then converted back to an electrical voltage by solving the indirect piezoelectric effect and is fed to the P-SPICE circuit simulation.

A general scheme for the full simulation can be seen presented schematically in Figure 4 a, a snapshot of the wave propagation at 7µs after signal excitation can be seen in Figure 4 b where a 270° revolved transducer with superimposed surface velocity is presented. To reduce computational intensity of the model the transducer was modeled in a

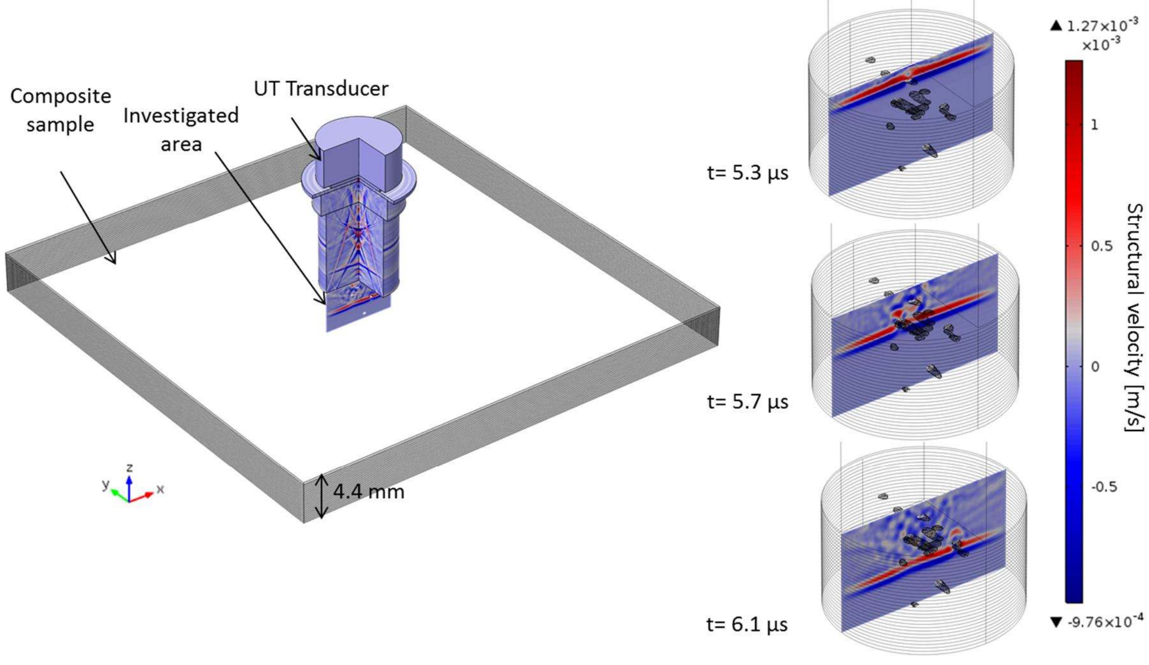
2D axisymmetric approach and was coupled to the 3D sample by a general extrusion-coupling operator.

For modeling a 5 MHz pulse propagation in an anisotropic media such as a multilayered composite the FEM code requires the use of a very high mesh resolution. Convergence studies for the mesh size and time step are necessary in order to validate the simulation work. For each type of material, the maximum element size of the mesh is defined to respect the $\lambda/5$ condition with λ being the shortest relevant wavelength in that material. For the composite material, a maximum element size of 0.12 mm and a time step of 2 ns were found to yield a convergent solution.

The final validation of the transducer modeling is done using a steel calibration block by directly comparing the experimental signal with the calculated one. The comparison of the A-scan pulses can be seen in Figure 4c. A good quantitative agreement was found between the voltage amplitudes of the signals, especially for the surface reflection and also the back wall echo.

3. Modelling ultrasound propagation in a composite structure with embedded defects

To model a composite material with embedded defects, the validated transducer model is now linked to a composite sample with artificial $[0/90_{14}]_{\text{sym}}$ layup with fiber axis along x-direction and embedded CT extracted defects as shown in Figure 5a. A sequence of images shows the wave propagation in the cross-section of the modeled composite sample using the structural velocity as color information. The presence of the embedded pores causes a scattering of the incident ultrasonic wave. Small reflections of the initial wave can be seen being reflected back to the sensor at 5.7 µs and 6.1 µs.



a.

b.

Fig. 5. a. Representation of the model configuration **b**. Interaction of the ultrasonic waves with the embedded pores at three distinct time steps.

Due to the computational intensity caused by the real geometry of the pores the question is whether these are required to be described in that level of detail. Thus, the following step is to compare the previous result with defect models for porosity described through artificially generated elliptical pores. Therefore, characteristic distributions of the pore size, shape and their orientation are extracted from the CT scans and are listed in Table 1. As can be seen in Figure 6a such artificial pores can be reasonably approximated as ellipsoids.

Table 1. Elliptical pores properties

Limits	**Semi-axis on x [mm]**	**Semi-axis on y [mm]**	**Semi-axis on z [mm]**
Max	0.205	1.03	0.14
Min	0.09	0.105	0.05

The artificial pores are built using a Matlab routine and are directly imported in Comsol Multiphysics by using the Matlab® Livelink^{TM} feature. Location of the pores is prescribed using a random function for generating coordinates within the sample volume. The porosity volume was chosen to correspond to the value detected by the Porosity/Inclusion Analysis Module in VG Studio MAX 2.1which is found to be 0.79mm³. The artificially generated pores can be seen presented as 3D geometry in Figure 7c.

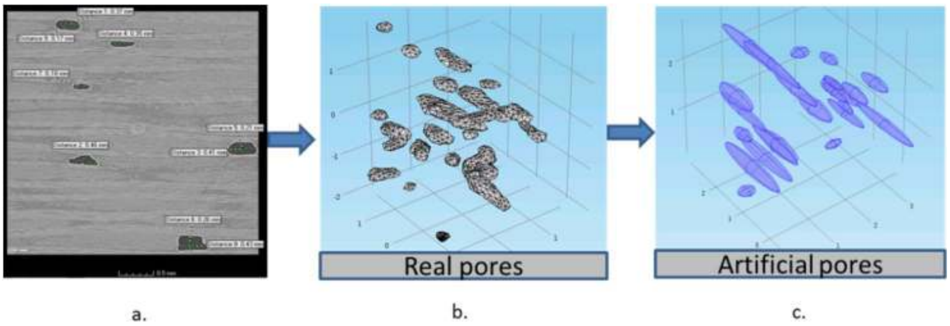


Fig. 6. a. Cross section of the CT-scan of the laminate **b**. Extracted pore geometry **c**. Artificial pore geometry.

The resulting calculated A-scans for both scenarios are presented in Figure 7. First, the A-Scan result of the CT extracted porosity is compared to a reference signal of a sample without any porosity. The decrease of the back wall echo due to the presence of the porosity is clearly seen.

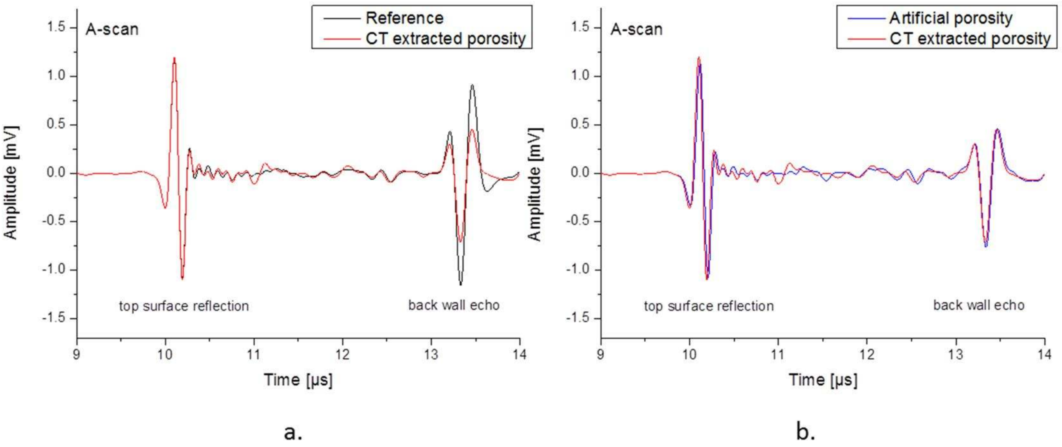


Fig. 7. a. Calculated signals for the samples with extracted pore geometries and free of pores (reference) **b**. Calculated signals for the samples with extracted pore geometries pores and artificial pore geometries.

In Figure 7b the calculated signals of CT extracted and artificial porosity are directly compared. Clearly, the signals are not identical in shape in between the sample top surface reflection and back wall echo. The small reflections are given by the reflections of specific pores which have different shape, magnitude and position in each sample as seen from

Figure 6. Nevertheless, the total contribution of the scatter at the pores is causing a decrease of the back wall echo magnitude. Here, the simulated signals obtained of CT extracted and artificial porosity show quantitatively good agreement.

4. Conclusions

With the present approach it has been demonstrated how finite element modeling can be used to calculate A-scan signals from ultrasonic testing. This simulation method included realistic geometries by explicit modeling of the sensor and test sample geometry. For the validation of the modeling method, experiments on a calibration steel block using the same ultrasonic sensor were conducted. The simulated ultrasonic signal obtained for the calibration block show good agreement to the experimental signal. Within this research, a work-flow to achieve geometry reconstruction from CT for the test laminate modeling it is also presented.

Based on these simulations it was demonstrated how ultrasonic signal propagation is affected by the porosity presence in the laminate sample. The back wall echo showed significant decay of magnitude with increasing porosity compared to the reference signal of a sample without any porosity. The change in magnitude of the back wall echo was found to be very small when using an artificial described geometry for the porosity. Also, the approach is generally applicable for similar investigations, i.e. to study other type of defects and their impact upon ultrasonic signal propagation/detection or to consider the effect of composite layers in the sample laminate.

Acknowledgments

The authors would like to thank Federal Ministry of Education and Research (BMBF) for the funding of the project MAIzfp within the leading edge cluster MAI Carbon.

References

[1] S. Lonné, L. de Roumilly, L. Le Ber, S. Mahaut and G. Cattiaux, *Experimental validation of Civa Ultrasonic Simulations*, 5th NDE, **2006**

[2] F. Foucher, P. Dubois, *Applications and recent evolutions of the CIVA Simulation Platform;* MENDT, **2012**

[3] R. Ludwig, W. Lord, *Finite-element study of ultrasonic wave propagation and scattering in an aluminum block*, Materials Evaluation, Vol. 46, pp. 156-161, **1988**.

[4] M. G. R. Sause, J. Scharrighausen, S. Horn, *Identification of the failure mechanisms in thermoplastic composites by acoustic emission measurmentss*, 19th International Conference on Composite Materials, **2013**.

[5] M. G. R. Sause, M. A. Hamstad, S. Horn, *Finite Element Modeling Of Conical Acoustic Emission Sensors And Corresponding Experiments*, Sensors and Actuators , *184(A)*, pp 64– 71, **2012**

[6] M. G. R. Sause and A.-M. Zelenyak, *Modellierung von Schallemissionsquellen auf Basis von volumetrischen Bildinformationen*, in 20. Kolloquium Schallemission, pp. 1–9, **2015**.

[7] D. R. Kiefel, B. Stoessel, C. Plank, J. Heinzl and Kastner, *CFRP porosity characterisation using µ-Computed Tomography with optimized test parameters supported by XCT-simulation,* Proceedings of iCT, **2014**.

[8] K. W. Pratt, *Digital image processing*, 3rd edition Willey, pp. 551-587, **2001**.