

# Numerical Vibration Analysis of Impacted CFRP Specimens Using the COMSOL Multiphysics® Software

P. Jatzlau<sup>1</sup>, F. Seybold<sup>1</sup>, C. Grosse<sup>1</sup>

<sup>1</sup>Technical University of Munich, Chair of Non-destructive Testing, Munich, Germany

## Abstract

Non-destructive testing of carbon fiber reinforced composites (CFRP) structures can be fairly time consuming and expensive. The question at hand is if defects in CFRP materials can be detected via modal testing in a first step to determine whether it is necessary to apply more thorough methods like ultrasound and x-ray. Therefore the intent of this work is to simulate how vibration modes are altered by defects resulting from an impact. In order to obtain a first insight into the vibration behavior of CFRP with and without defects, simple specimens have been modeled with COMSOL Multiphysics® software. The specimens are rectangular laminates with different layups. In this process the Structural Mechanics Module with the Solid Mechanics interface have been used to conduct an Eigenfrequency Study. Different parameters such as fiber angle or geometrical dimensions have been adjusted to give accurate results which were validated by experimental examination of the specimens. In the further process, the specimens were impacted to yield the desired defects. The defects are currently modeled in COMSOL®. It is planned to investigate the size of the defect and alter the material stiffness in this region of the COMSOL model.

The results of the simulation of the specimens without defects yield eigenfrequencies that are accurate within about 5% over a broad frequency range, compared to the experimental results. The results of the simulation of the impacted specimens will show if the changes measured experimentally can also be observed in the COMSOL model. A conclusion cannot be given at this stage of the process but a short outlook will be provided. If it shows that the alteration of vibration can be simulated, it is planned to do a frequency response analysis and simulate the alteration of acoustic emissions of the specimens due to defects. COMSOL will be used for this task because of its ability to couple the Structural Mechanics Module and the Acoustics Module.