The aim of the second and final part of this study is to simulate fatigue crack growth in the tested Sandwich Tear Test specimens, described in Part I, using the finite element method. To accelerate the simulation, a cycle jump method is utilized and implemented in the finite element routine. The proposed method is based on conducting finite element analysis for a set of cycles to establish a trend line, extrapolating the trend line spanning many cycles, and use the extrapolated state as initial state for additional finite element simulations. The measured da/dN relations of the face/core interface in Part I are used as input to the fatigue crack growth simulation routine. A fair accuracy with 99% saving in computation time is achieved in the simulation of the Sandwich Tear Test specimens with H100 core. However, for the Sandwich Tear Test specimens with H45 core a large deviation between the numerical and experimental results is seen due to a partial unstable fatigue crack growth which was not taken into account in the simulations.