POSTBUCKLING ANALYSIS OF FIBRE COMPOSITE STRINGER STIFFENED PANELS – DEVELOPMENT OF A FAST DESIGN TOOL

Alexander Kling, Richard Degenhardt, Rolf Zimmermann

DLR, Institute of Structural Mechanics, Braunschweig, Germany

As a result of consecutive reductions within development times of future composite aircraft structures, there is a need of a reliable simulation procedure for post-buckling behavior of stringer stiffened fibre reinforced composite panels, which is faster by an order of magnitude than respective nonlinear finite element simulations at an accuracy satisfying design requirements. The POSICOSS project deals with this demand.

DLR's simulation procedure considers an ideal stringer-skin connection, pure axial, pure shear and combined load cases, initial geometric imperfections and laminates made of unidirectional orthotropic prepreg material. The requested output of the analysis is the axial stiffness (e.g. load-shortening) in the pre- and postbuckling region and the deformed structure up to the onset of degradation. Additionally, a survey of bifurcation and limit points will be provided. The basic idea of the fast tool – to reduce the number of degrees of freedom significantly – can be best characterized as a hybrid reduced basis technique, which is clarified in Figure 1.

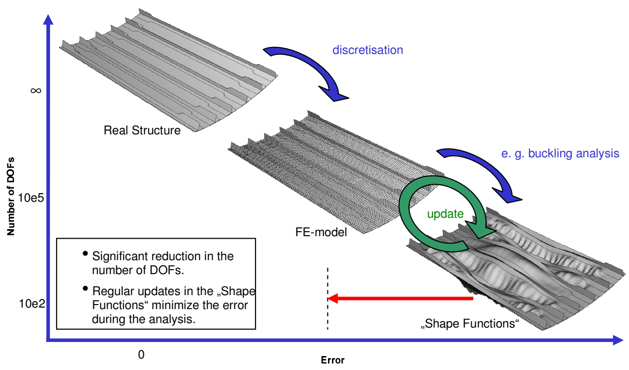

Figure 1: Basic idea of the concept

The real composite panel (with an infinite number of degrees of freedom, DOFs) will be discretized, to obtain a conventional finite element model. The horizontal axis depicts the increasing error due to the reduction in the number of degrees of freedoms (vertical axis). This finite element model will be used to extract a small number of "shape functions" (e.g. buckling modes), which can be utilized subsequently to analyze the structural behavior. The shape functions will be updated regularly, based on a predetermined error limit, to restrict the error during the nonlinear analysis. Therefore error sensing and error control will be an important part during the calculation.

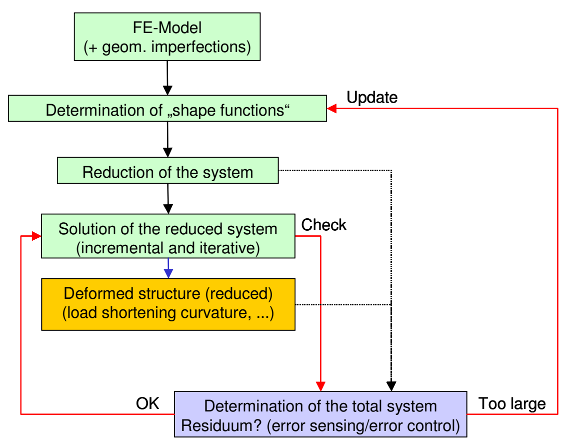

Figure 2: Algorithm of the concept

The algorithm, as shown in Figure 2, transfers the basic idea of the concept into a programmable software tool. The analysis starts with a conventional finite element model (e.g. generated by a commercial preprocessor) of the examined structure. Initial geometric imperfections can be superposed before the analysis starts, which is comparable to the approach implemented in commercial finite element analysis tools like ABAQUS. In a next step the number of DOFs will be reduced with a limited number of shape functions. Subsequently, the reduced system will be solved using a conventional incremental/iterative solution procedure to obtain the deformed structure as well as the load-shortening curve.

Some preliminary results of simple beam structures in the prototyping environment showed the potential of the concept with respect to shorter computational time and appropriate accuracy of the results for design purposes. B2000, an open source finite element program, will be used to implement the algorithm for shell-type structures.

In the presentation the basic concept and its transfer to the algorithm will be presented in detail, and preliminary results will be given.