Peter Linde

See: http://www.linkedin.com/pub/peter-linde/a/902/291

2002 – present at Airbus:
Research & Technology - Airframe Architecture & Integration at Airbus, Hamburg area, Germany

Education:
1995 – 1998 Research Fellow at the University of California, Los Angeles (M.S. degree)
Stanford University (Eng. Degree in Structural Mechanics)
Eidgenössische Technische Hochschule (ETH), Zürich (Ph.D. degree)

Areas of interest:
Aviation, Structural mechanics, Aeronautics and space research, Composites: thermoplastics, nano-composites

Awards:
Airbus Award of Excellence in Innovation, 2006

Selected Publications:
T. Möcker (1), P. Linde (3), S. Kraschin (2), F. Goetz (1), J. Marsolek (1), W. Wohlers (3)
(1) Abaqus Deutschland GmbH, 52062 Aachen, Germany, E-mail: Torsten.MOECKER@3ds.com
(2) Bishop GmbH - Aeronautical Engineers, 22587 Hamburg, Germany
(3) Airbus Deutschland GmbH, 21129 Hamburg, Germany
"ABAQUS FEM Analysis of the Postbuckling Behaviour of Composite Shell Structures", (publisher, year not given; most recent reference is 2008)

ABSTRACT: For the design of stiffened composite shell structures the knowledge of the structural response in the postbuckling region is an important topic. Accordingly, tools are required that enable an accurate and reliable prediction of the postbuckling behaviour. In this paper it is shown how the finite element code Abaqus can be used for this purpose. When performing finite element simulations, a large amount of time is often needed to build up the finite element model - in particular if the model consists of several parts with complex geometries. For this reason the preprocessing tool Abaqus/CAE provides an interface which allows the user to automate repetitive tasks. Based on this interface, a tool simplifying the pre- and postprocessing of shell structures stiffened by stringers and frames was developed by Abaqus Deutschland for the company Airbus. Next to a summary of the abilities of this tool, the main focus of this paper is on discussing several modelling techniques that are used to enable a realistic idealisation of the physical problem and on presenting simulation results for an exemplary structure. Based on this example, the influence of modelling details like mesh density and geometric imperfections on the prediction of the failure load is discussed.