The 8th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2022
5 – 9 June 2022, Oslo, Norway

A method for communication between user materials during runtime in Abaqus®

P. Rose[1,2]*, L. Münch[3], M. Linke[1] and P. Middendorf[3]

[1] Hamburg University of Applied Sciences, Department of Automotive & Aeronautical Engineering, Berliner Tor 9, 20099 Hamburg, Philip.Rose@HAW-Hamburg.de, https://www.haw-hamburg.de
[2] Universitat Politècnica de València, Camino de Vera, s/n. 46022 - Valencia, http://www.upv.es/es
[3] University of Stuttgart, Paffenwaldring 31, 70569 Stuttgart, lmuench@IFB.Uni-Stuttgart.de, https://www.ifb.uni-stuttgart.de

Key Words: *User-Subroutine, Communication during run time, Finite Element (FE)*

The application of the Finite Element Method (FEM) has developed considerably in recent decades. While in the early days of the FEM only linear-elastic material models were available in commercial Finite Element (FE) programs, today non-linear and also damage-considering material models are offered. But even these are often not capable of correctly representing the complex material behaviour of modern composite materials. Therefore, FE programs often provide the users with the option of integrating their own material models into the simulation.

These programs, often called "user subroutines" or "user materials", represent an intensively researched area in the simulation of material behaviour, which is reflected in the large number of publications on such developments (exemplarily see [1, 2]). These material models often require a large amount of data values if they are introduced within the simulation model by finite elements [2]. Especially in the field of composites there is a need to use a separate "user material" for each constituent material. Since there is a mutual interaction between the components in real composites, it is obvious that this interaction must also be represented between the material models within a simulation in order to make accurate predictions about the material behaviour. To enable such an interaction between different material models in a simulation, a communication during runtime is required in which additionally needed data from the surrounding elements is exchanged between the user materials.

In this paper, a method is presented to enable such information exchange during simulation runtime in the commercial FE software ABAQUS/CAE 2019 (Dassault Systèmes, Vélizy-Villacoublay, France) using external databases as well as structured global arrays and some built-in functions of the software. This allows the simultaneous application of several advanced user material models within one simulation and enable them to communicate during runtime, resulting in the possibility of making high accurate simulations of composite materials.

REFERENCES

[1] B.L.V. Bak, A. Turon, E. Lindgaard, E. Lund, A benchmark study of simulation methods for high-cycle fatigue-driven delamination based on cohesive zone models, *Composite Structures* Vol. **164**, *pp. 198–206,* Elsevier, 2017

[2] R. Sachse, A. K. Pickett, W. Essig, P. Middendorf, Experimental and numerical investigation of the influence of rivetless nut plate joints on fatigue crack growth in adhesively bonded composite joints. *International Journal of Fatigue* Vol. **105**, pp. 262–275, 2017.