Bachelor-, Studien-, Seminar- oder Masterarbeit

Abstract modelling of annular composite disc in Abaqus using python scripting and user defined element.

Description

In the realm of modern engineering, the utilization of large composite structures is increasingly prevalent, especially in industries such as aerospace, automotive, and civil engineering. These structures are commonly joined using bolts, a method that necessitates precise and reliable simulation techniques to ensure structural integrity and safety. However, the finite element simulation of bolted joints in composite materials presents significant challenges. One of the primary issues is the extensive computational time required to accurately simulate a single composite joint. This complexity is compounded when attempting to model large structures featuring multiple bolted joints, often referred to as a field of bolts. The project aims to increase computational efficiency in simulation of joints using abstract models supplemented with Machine learning. The purpose of this thesis is to create an abstract model of an annular composite disc using beam elements, based on classical laminate theory (CLT). The performance of the abstract model will then be compared with a high-fidelity model. Students can gain valuable experience in developing a user defined element (.uel) and Python scripting.

Tasks

- Python scripting to create Abstract model in Abaqus
- Modelling and Stress Analysis in Abaqus
- Fortran programming to develop user element

Prerequisites

We are seeking a driven student pursuing a degree in Mechanical Engineering, Civil Engineering, Computational Sciences, or a related field. The candidate should possess foundational knowledge of Finite Element Method (FEM) and composites, particularly CFRP. Prior experience with Abaqus and basic Python programming skills are preferred.

Ansprechpartner

Aditya Bansod, M. Sc.
Institut für Statik und Dynamik
Appelstr. 9A
30167 Hannover
Tel.: 0511-762-17426
E-Mail an: a.bansod@isd.uni-hannover.de